CVNC™-M2
User Guide

CADDS® 5i Release 14

DOC38514-010

Parametric Technology Corporation

Copyright © 2005 Parametric Technology Corporation. All Rights Reserved.
User and training documentation from Parametric Technology Corporation (PTC) is subject to the copyright laws of the United States and other countries and is provided under a license agreement that restricts copying, disclosure, and use of such documentation. PTC hereby grants to the licensed user the right to make copies in printed form of this documentation if provided on software media, but only for internal/personal use and in accordance with the license agreement under which the applicable software is licensed. Any copy made shall include the PTC copyright notice and any other proprietary notice provided by PTC. This documentation may not be disclosed, transferred, modified, or reduced to any form, including electronic media, or transmitted or made publicly available by any means without the prior written consent of PTC and no authorization is granted to make copies for such purposes.

Information described herein is furnished for general information only, is subject to change without notice, and should not be construed as a warranty or commitment by PTC. PTC assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the United States and other countries. It may not be copied or distributed in any form or medium, disclosed to third parties, or used in any manner not provided for in the software licenses agreement except with written prior approval from PTC. UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION.

Registered Trademarks of Parametric Technology Corporation or a Subsidiary
Advanced Surface Design, Behavioral Modeling, CADDS, Computervision, CounterPart, Create · Collaborate · Control, EPD, EPD.Connect, Expert Machinist, Flexible Engineering, GRANITE, HARNESSDESIGN, Info*Engine, InPart, MECHANICA, Optegra, Parametric Technology, Parametric Technology Corporation, PartSpeak, PHOTORENDER, Pro/DESKTOP, Pro/E, Pro/ENGINEER, Pro/HELP, Pro/INTRALINK, Pro/MECHANICA, Pro/TOOLKIT, Product First, Product Development Means Business, Product Makes the Company, PTC, the PTC logo, PT/Products, Shaping Innovation, Simple · Powerful · Connected, The Way to Product First, and Windchill.

Trademarks of Parametric Technology Corporation or a Subsidiary
3DPAINT, Associative Topology Bus, AutobuildZ, CDRS, CV, CVact, CVaec, CVdesign, CV-DORS, CVMAC, CVNC, CVToolmaker, EDAcompare, EDAconduit, DataDoctor, DesignSuite, DIMENSION III, Distributed Services Manger, DIVISION, e/ENGINEER, eNC Explorer, Expert Framework, Expert MoldBase, Expert Toolmaker, FlexPDM, FlexPLM, Harmony, InterComm, InterComm Expert, InterComm EDAcompare, InterComm EDAconduit, ISSM, KDiP, Knowledge Discipline in Practice, Knowledge System Driver, ModelCHECK, MoldShop, NC Builder, POLYCAPP, Pro/ANIMATE, Pro/ASSEMBLY, Pro/CABLING, Pro/CASTING, Pro/CDT, Pro/CMM, Pro/COLLABORATE, Pro/COMPOSITE, Pro/CONCEPT, Pro/CONVERT, Pro/DATA for PDGS, Pro/DESIGNER, Pro/DETAIL, Pro/DIAGRAM, Pro/DIEFACE, Pro/DRAW, Pro/ECAD, Pro/ENGINE, Pro/FEATURE, Pro/FEM-POST, Pro/FICIENCY, Pro/FLY-THROUGH, Pro/HARNESS, Pro/INTERFACE, Pro/LANGUAGE, Pro/LEGACY, Pro/LIBRARYACCESS, Pro/MESH, Pro/Model.View, Pro/MOLDESIGN, Pro/NC-ADVANCED, Pro/NC-CHECK, Pro/NC-MILL, Pro/NC-POST, Pro/NC-SHEETMETAL, Pro/NC-TURN, Pro/NC-WEDM, Pro/NC-Wire EDM, Pro/NETWORK ANIMATOR, Pro/NOTEBOOK, Pro/PDM, Pro/PHOTORENDER, Pro/PIPING, Pro/PLASTIC ADVISOR, Pro/PLOT, Pro/POWER DESIGN, Pro/PROCESS, Pro/REPORT, Pro/REVIEW, Pro/SCAN-TOOLS, Pro/SHEETMETAL, Pro/SURFACE, Pro/VERIFY, Pro/Web.Link, Pro/Web.Publish, Pro/WELDING, ProductView, PTC Precision, Routed Systems Designer, Shrinkwrap, The Product Development Company, Validation Manager, Wildfire, Windchill DynamicDesignLink, Windchill PartsLink, Windchill PDMLink, Windchill ProjectLink, and Windchill SupplyLink.

Patents of Parametric Technology Corporation or a Subsidiary
Registration numbers and issue dates follow. Additionally, equivalent patents may be issued or pending outside of the United States. Contact PTC for further information.

GB2366639B 13-October-2004	GB2353376 05-November-2003	5,140,321 18-August-1992
GB2363208 25-August-2004	GB2354686 15-October-2003	5,423,023 05-June-1990
(EP/DE/GB)0812447 26-May-2004	6,545,671 B1 08-April-2003	4,310,615 21-December-1998
GB2365567 10-March-2004	GB2354685B 18-June-2003	4,310,614 30-April-1996

(GB)2388003B 21-January-2004	GB2354683B 04-June-2003	4,310,614 22-April-1999
6,665,569 B1 16-December-2003	6,608,623 B1 19-August-2003	5,297,053 22-March-1994
GB2353115 10-December-2003	6,473,673 B1 29-October-2002	5,513,316 30-April-1996
6,625,607 B1 23-September-2003	GB2354683B 04-June-2003	5,689,711 18-November-1997
6,580,428 B1 17-June-2003	6,447,223 B1 10-September-2002	5,506,950 09-April-1996
GB2354684B 02-July-2003	6,308,144 23-October-2001	5,428,772 27-June-1995
GB2384125 15-October-2003	5,680,523 21-October-1997	5,850,535 15-December-1998
GB2354096 12-November-2003	5,838,331 17-November-1998	5,557,176 09-November-1996
GB2354924 24-September-2003	4,956,771 11-September-1990	5,561,747 01-October-1996
6,608,623 B1 19-August-2003	5,058,000 15-October-1991	(EP)0240557 02-October-1986

Third-Party Trademarks

Adobe, Acrobat, Distiller, and the Acrobat logo are trademarks of Adobe Systems Incorporated. Advanced ClusterProven, ClusterProven, and the ClusterProven design are trademarks or registered trademarks of International Business Machines Corporation in the United States and other countries and are used under license. IBM Corporation does not warrant and is not responsible for the operation of this software product. AIX is a registered trademark of IBM Corporation. Allegro, Cadence, and Concept are registered trademarks of Cadence Design Systems, Inc. Apple, Mac, Mac OS, and Panther are trademarks or registered trademarks of Apple Computer, Inc. AutoCAD and Autodesk Inventor are registered trademarks of Autodesk, Inc. Baan is a registered trademark of Baan Company. CADAM and CATIA are registered trademarks of Dassault Systemes. COACH is a trademark of CADTRAIN, Inc. CYA, iArchive, HOTbackup, and Virtual StandBy are trademarks or registered trademarks of CYA Technologies, Inc. DOORS is a registered trademark of Telelogic AB. FLEXlm and FLEXnet are registered trademarks of Macrovision Corporation. Geomagic is a registered trademark of Raindrop Geomagic, Inc. EVERSYNC, GROOVE, GROOVEFEST, GROOVE.NET, GROOVE NETWORKS, iGROOVE, PEERWARE, and the interlocking circles logo are trademarks of Groove Networks, Inc. Helix is a trademark of Microcadam, Inc. HOOPS is a trademark of Tech Soft America, Inc. HP-UX is a registered trademark of Hewlett-Packard Company. I-DEAS, Metaphase, Parasolid, SHERPA, Solid Edge, TeamCenter, UG-NX, and Unigraphics are trademarks or registered trademarks of UGS Corp. InstallShield is a registered trademark and service mark of InstallShield Software Corporation in the United States and/or other countries. Intel is a registered trademark of Intel Corporation. IRIX is a registered trademark of Silicon Graphics, Inc. I-Run and ISOGEN are registered trademarks of Alias Ltd. LINUX is a registered trademark of Linus Torvalds. MainWin and Mainsoft are trademarks of Mainsoft Corporation. MatrixOne is a trademark of MatrixOne, Inc. Mentor Graphics and Board Station are registered trademarks and 3D Design, AMPLE, and Design Manager are trademarks of Mentor Graphics Corporation. MEDUSA and STHENO are trademarks of CAD Schroer GmbH. Microsoft, Microsoft Project, Windows, the Windows logo, Windows NT, Windows XP, Visual Basic, and the Visual Basic logo are registered trademarks of Microsoft Corporation in the United States and/or other countries. Moldflow is a registered trademark of Moldflow Corporation. Netscape and the Netscape N and Ship's Wheel logos are registered trademarks of Netscape Communications Corporation in the U.S. and other countries. Oracle is a registered trademark of Oracle Corporation. OrbixWeb is a registered trademark of IONA Technologies PLC. PDGS is a registered trademark of Ford Motor Company. RAND is a trademark of RAND Worldwide. Rational Rose is a registered trademark of Rational Software Corporation. RetrievalWare is a registered trademark of Convera Corporation. RosettaNet is a trademark and Partner Interface Process and PIP are registered trademarks of RosettaNet, a nonprofit organization. SAP and R/3 are registered trademarks of SAP AG Germany. SolidWorks is a registered trademark of SolidWorks Corporation. All SPARC trademarks are used under license and are trademarks or registered trademarks of SPARC International, Inc. in the United States and in other countries. Products bearing SPARC trademarks are based upon an architecture developed by Sun Microsystems, Inc. Sun, Sun Microsystems, the Sun logo, Solaris, UltraSPARC, Java and all Java based marks, and "The Network is the Computer" are trademarks or registered trademarks of Sun Microsystems, Inc. in the United States and in other countries. 3Dconnexion is a registered trademark of Logitech International S.A. TIBCO is a registered trademark and TIBCO ActiveEnterprise, TIBCO Designer, TIBCO Enterprise Message Service, TIBCO Rendezvous, TIBCO TurboXML, and TIBCO BusinessWorks are the trademarks or registered trademarks of TIBCO Software Inc. in the United States and other countries. WebEx is a trademark of WebEx Communications, Inc.

Third-Party Technology Information

Certain PTC software products contain licensed third-party technology:

Rational Rose 2000E is copyrighted software of Rational Software Corporation.

RetrievalWare is copyrighted software of Convera Corporation.

VisTools library is copyrighted software of Visual Kinematics, Inc. (VKI) containing confidential trade secret information belonging to VKI.

HOOPS graphics system is a proprietary software product of, and is copyrighted by, Tech Soft America, Inc.

I-Run and ISOGEN are copyrighted software of Alias Ltd.

Xdriver is copyrighted software of 3Dconnexion, Inc, a Logitech International S.A. company.

G-POST is copyrighted software and a registered trademark of Intercim.

VERICUT is copyrighted software and a registered trademark of CGTech.

FLEXnet Publisher is copyrighted software of Macrovision Corporation.

Pro/PLASTIC ADVISOR is powered by Moldflow technology.

MainWin Dedicated Libraries are copyrighted software of Mainsoft Corporation.

DFORMD.DLL is copyrighted software from Compaq Computer Corporation and may not be distributed.

LightWork Libraries are copyrighted by LightWork Design 1990-2001.

Visual Basic for Applications and Internet Explorer is copyrighted software of Microsoft Corporation.

Parasolid is © UGS Corp.

TECHNOMATIX is copyrighted software and contains proprietary information of Technomatix Technologies Ltd.

TIBCO ActiveEnterprise, TIBCO Designer, TIBCO Enterprise Message Service, TIBCO Rendezvous, TIBCO TurboXML, and TIBCO BusinessWorks are provided by TIBCO Software Inc.

Technology "Powered by Groove" is provided by Groove Networks, Inc.

Technology "Powered by WebEx" is provided by WebEx Communications, Inc.

Oracle 8i run-time, Oracle 9i run-time, and Oracle 10g run-time are Copyright © 2002-2004 Oracle Corporation. Oracle programs provided herein are subject to a restricted use license and can only be used in conjunction with the PTC software they are provided with.

Adobe Acrobat Reader and Adobe Distiller are copyrighted software of Adobe Systems Inc. and are subject to the Adobe End-User License Agreement as provided by Adobe with those products.

METIS, developed by George Karypis and Vipin Kumar at the University of Minnesota, can be researched at http://www.cs.umn.edu/~karypis/metis. METIS is © 1997 Regents of the University of Minnesota.

Windchill Info*Engine Server contains IBM XML Parser for Java Edition and the IBM Lotus XSL Edition.

Pop-up calendar components Copyright © 1998 Netscape Communications Corporation. All Rights Reserved.

Apache Server, Tomcat, Xalan, Xerces, and Jakarta are technologies developed by, and are copyrighted software of, the Apache Software Foundation (http://www.apache.org) – their use is subject to the terms and limitations at: http://www.apache.org.

UnZip (© 1990-2001 Info-ZIP, All Rights Reserved) is provided "AS IS" and WITHOUT WARRANTY OF ANY KIND. For the complete Info-ZIP license see http://www.info-zip.org/doc/LICENSE.

The Java™ Telnet Applet (StatusPeer.java, TelnetIO.java, TelnetWrapper.java, TimedOutException.java), Copyright © 1996, 97 Mattias L. Jugel, Marcus Meißner, is redistributed under the GNU General Public License. This license is from the original copyright holder and the Applet is provided WITHOUT WARRANTY OF ANY KIND. You may obtain a copy of the source code for the Applet at http://www.mud.de/se/jta (for a charge of no more than the cost of physically performing the source distribution), by sending e-mail to leo@mud.de or marcus@mud.de—you are allowed to choose either distribution method. Said source code is likewise provided under the GNU General Public License.

GTK+ - The GIMP Toolkit is licensed under the GNU Library General Public License (LGPL). You may obtain a copy of the source code at http://www.gtk.org, which is likewise provided under the GNU LGPL.

zlib software Copyright © 1995-2002 Jean-loup Gailly and Mark Adler.

OmniORB is distributed under the terms and conditions of the GNU General Public License and GNU Library General Public License.

The Java Getopt.jar file, copyright 1987-1997 Free Software Foundation, Inc.

Java Port copyright 1998 by Aaron M. Renn (arenn@urbanophile.com), is redistributed under the GNU LGPL. You may obtain a copy of the source code at http://www.urbanophile.com/arenn/hacking/download.html. The source code is likewise provided under the GNU LGPL.

CUP Parser Generator Copyright ©1996-1999 by Scott Hudson, Frank Flannery, C. Scott Ananian–used by permission. The authors and their employers disclaim all warranties with regard to this software, including all implied warranties of merchantability and fitness. In no event shall the authors or their employers be liable for any special, indirect or consequential damages, or any damages whatsoever resulting from loss of use, data or profits, whether in an action of contract, negligence or other tortious action arising out of or in connection with the use or performance of this software.

This product may include software developed by the OpenSSL Project for use in the OpenSSL Toolkit. (http://www.openssl.org): Copyright © 1998-2003 The OpenSSL Project. All rights reserved. This product may include cryptographic software written by Eric Young (eay@cryptsoft.com).

ImageMagick software is Copyright © 1999-2005 ImageMagick Studio LLC, a nonprofit organization dedicated to making software imaging solutions freely available. ImageMagick is freely available without charge and provided pursuant to the following license agreement: http://www.imagemagick.org/script/license.php.

Gecko and Mozilla components are subject to the Mozilla Public License Version 1.1 at http://www.mozilla.org/MPL. Software distributed under the Mozilla Public License (MPL) is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either expressed or implied. See the MPL for the specific language governing rights and limitations.

Mozilla Japanese localization components are subject to the Netscape Public License Version 1.1 (at http://www.mozilla.org/NPL). Software distributed under the Netscape Public License (NPL) is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either expressed or implied (see the NPL for the rights and limitations that are governing different languages). The Original Code is Mozilla Communicator client code, released March 31, 1998 and the Initial Developer of the Original Code is Netscape Communications Corporation. Portions created by Netscape are Copyright © 1998 Netscape Communications Corporation. All Rights Reserved. Contributors: Kazu Yamamoto (kazu@mozilla.gr.jp), Ryoichi Furukawa (furu@mozilla.gr.jp), Tsukasa Maruyama (mal@mozilla.gr.jp), Teiji Matsuba (matsuba@dream.com).

iCal4j is Copyright © 2005, Ben Fortuna, All rights reserved. Redistribution and use of iCal4j in source and binary forms, with or without modification, are permitted provided that the following conditions are met: (i) Redistributions of source code must retain the above copyright notice, this list of conditions, and the following disclaimer; (ii) Redistributions in binary form must reproduce the above copyright notice, this list of conditions, and the following disclaimer in the documentation and/or other materials provided with the distribution; and (iii) Neither the name of Ben Fortuna nor the names of any other contributors may be used to endorse or promote products derived from this software without specific prior written permission. iCal4j SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Software may contain the Independent JPEG Group's JPEG software. This software is Copyright © 1991-1998, Thomas G. Lane. All Rights Reserved. This software is based in part on the work of the Independent JPEG Group.

Software may contain libpng, Copyright © 2004 Glenn Randers-Pehrson, which is distributed according to the disclaimer and license (as well as the list of Contributing Authors) at http://www.libpng.org/pub/png/src/libpng-LICENSE.txt.

UNITED STATES GOVERNMENT RESTRICTED RIGHTS LEGEND
This document and the software described herein are Commercial Computer Documentation and Software, pursuant to FAR 12.212(a)-(b) (OCT'95) or DFARS 227.7202-1(a) and 227.7202-3(a) (JUN'95), and are provided to the US Government under a limited commercial license only. For procurements predating the above clauses, use, duplication, or disclosure by the Government is subject to the restrictions set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software Clause at DFARS 252.227-7013 (OCT'88) or Commercial Computer Software-Restricted Rights at FAR 52.227-19(c)(1)-(2) (JUN'87), as applicable. 090805

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA

Table of Contents

Preface

Related Documents ____ xv
Book Conventions ____ xvi
Window Managers and the User Interface ____ xvii
Online User Documentation ____ xvii
Online Command Help ____ xviii
Printing Documentation ____ xviii
Resources and Services ____ xix
Documentation Comments ____ xix

Overview of CVNC-M2

Purpose of CVNC-M2 ____ 1-2
Tasks Performed by CVNC-M2 ____ 1-2
CVNC-M2 Features ____ 1-4
Job Control Files ____ 1-4
System Variables ____ 1-4
System Monitoring Tools ____ 1-5
Macros ____ 1-5
Command Files ____ 1-5
Output Formats ____ 1-5
Output Pass-through Statements ____ 1-6
How to Enter CVNC-M2 ____ 1-7
What CVNC-M2 Commands Do ____ 1-8

Base System Commands 1-8
CVNC-M2 Commands 1-8
Job Setup Commands 1-9
Operation Setup Commands 1-9
Motion Commands 1-10
How to Generate and Edit a Job Control File 1-11
Generating a JCF 1-11
Editing Data in a JCF 1-11

Job Setup: Machine Configuration

Initial Setup: Overview 2-2
Defining the Machine Tool Linear Coordinate System and Part Program Zero — DATUM 2-3
Specifying Home Location — HOMEPT 2-5
Using HOMEPT Before a Tool Change 2-6
Defining Milling Machines with Rotary Axes — CONFIG 2-8
The Relationship of the DATUM Cplane to Rotary Axes 2-9
Configuring Linear Axes: Overview 2-10
Defining the Travel Limits for Linear Motion 2-10
Validating Motion 2-10
Unsetting the Limits 2-10
Configuring 4-Axis Milling Machines: Overview 2-11
The Relationship of HOMEPT with CONFIG 2-11
Defining the Travel Limits for Rotational Motion 2-13
Resolving Multiple Solutions 2-13
Specifying the Direction of Rotation 2-14
Configuring a 4-Axis Milling Machine with a Rotary Head 2-15
Configuring an A-Axis Head Machine 2-15
Configuring a B-Axis Head Machine 2-16
Configuring a 4-Axis Milling Machine with a Rotary Table 2-17
AAXIS Table Machine 2-18
BAXIS Table Machine 2-19
C-Axis Table Machine 2-22

Configuring 5-Axis Milling Machines: Overview ____ 2-23
Configuring 5-Axis Milling Machines with Compound Heads____ 2-23
Compound Head: CAXIS and AAXIS Rotation ____ 2-23
Compound Head: C- and B-Axis Rotation____ 2-25
Configuring 5-Axis Milling Machines with Compound Tables____ 2-26
Configuring 5-Axis Milling Machines: Head and Table____ 2-29
Using the HZERO Modifier in Configuring Rotary Heads____ 2-30
Characteristics of the CONFIG Command ____ 2-32
Invalid Configurations ____ 2-33
Unsupported Configurations ____ 2-33

Job Setup: Tool Definition

Tool Definition Overview ____ 3-2
Setting Up Tool Libraries — TLIB ____ 3-3
Defining Tools — DEFTOOL ____ 3-4
Defining a Countersink Tool ____ 3-8
Defining 2-Parameter Tools with the MILL7 Modifier ____ 3-9
Listing and Deleting Tools — LISTOOL and DELTOOL____ 3-12
LISTOOL____ 3-12
DELTOOL____ 3-12

Operation Setup: Regulating Cutting Operations

Overview of Regulating Cutting Operations____ 4-2
Changing Tools — CHGTOOL ____ 4-3
Specifying a New Construction Plane — CPL____ 4-4
Selecting or Creating a New Cplane ____ 4-5
Restricting Cplane Selection to a Constant Z____ 4-6
Programming Rotary Axes — INDEX____ 4-8
Specifying Axes Directly____ 4-10

Using a Cplane to Define Orientation 4-13
Using a Normal to Align the Tool Axis 4-15
Using a Plane 4-15
System-generated Cplanes 4-15
Validating INDEX 4-18
Fully Aligning Rotary Axes with the Cplane — NOOPTIM Modifier 4-18
Specifying Tool Axis Only Normal to the Cplane — OPTIM Modifier 4-20
CLFile ORIGIN Statement Calculation-Rotary Tables 4-21
Alternatives when Indexing to a Cplane for a 5-Axis Machine 4-24
Using the CPL Command with Respect to INDEX 4-27
The Affect of MULTAX ON 4-28
Input to Output Coordinate System Relationships 4-29
Rotary Head Machine Tool 4-29
Rotary Table Machine Tool 4-31
CVNC to NC Machine Relationships 4-33
BAXIS Table Device 4-33
AAXIS Head Device 4-35

Operation Setup: Operational Parameters

Overview of Setting Operational Parameters 5-2
Reference Planes 5-2
Speed and Feed Rates 5-2
Coolant 5-2
Stock and Tolerance Values 5-2
Diameter Compensation Register 5-3
Setting Z-Planes — PLANE 5-4
Z-Plane Defaults 5-5
Specifying Modal Feed Rates — FEED 5-6
Feed Rate Defaults 5-6
Slowdown Feed Rates 5-7

Sharp Corners ____ 5-7
Acceleration and Deceleration ____ 5-8
Arc Motion ____ 5-9
Specifying Spindle Speed — SPEED ____ 5-11
Turning Coolant On and Off — COOLANT ____ 5-12
Setting Stock Offsets — STOCK ____ 5-13
Specifying Fillet Treatment ____ 5-16
Conditions when a Fillet Is Floated ____ 5-17
Conditions when a Fillet Is Not Floated ____ 5-19
Specifying Tolerance — TOLER ____ 5-21
Setting the Diameter Compensation Register — DIACOMP ____ 5-22
Contact Point Output ____ 5-22
Example (3-axis Tool path Output) ____ 5-23
Example (5-axis Tool path Output) ____ 5-24
Calculating Tool Offset — CALCRAD ____ 5-26
Multiaxis Output — MULTAX ____ 5-29

Tool Motion Generation: Milling

Overview of Milling Commands ____ 6-2
Moving the Cutter with PLUNGE and CUT ____ 6-2
Rough Cutting and Finishing with PROFILE and POCKET ____ 6-2
Machining Profiles and Pockets with Macros ____ 6-3
Area Clearance ____ 6-3
Cutting with Linear or Circular Motion — CUT ____ 6-4
Generating Circular Interpolation — CUT ARC ____ 6-11
Moving the Tool Along an Entity — CUT ENTITY ____ 6-12
Working with the CUT Commands ____ 6-17
CUT ON - Single Point of Normalcy ____ 6-17
CUT ON - Multiple Points of Normalcy ____ 6-17
CUT ON - No Points of Normalcy ____ 6-18
CUT TO/CUT PAST - Determining Side-of-Boundary ____ 6-19
Bias Points ____ 6-19

CUT ENTITY _____ 6-20
CUT CHECK ON - One Intersection Point _____ 6-20
CUT CHECK ON - Multiple Intersection Points _____ 6-21
CUT CHECK ON - No Intersection Points _____ 6-21
CUT CHECK TO/PAST - Determining Side-of-Boundary _____ 6-22
CUT CHECK _____ 6-22

Creating Plunging Motion — PLUNGE _____ 6-24

Machining Around Contiguous Entities — PROFILE _____ 6-25
Machining Several Profiles: Depth Value — DPROF Macro _____ 6-29
Machining Several Profiles: Stock Value — SPROF Macro _____ 6-30

Machining Within a Closed Boundary — POCKET _____ 6-31
Pocketing a Pinched-off Area _____ 6-32
Example 1 _____ 6-33
Example 2 _____ 6-33
Example 3 _____ 6-34
Pocketing with Multiple Passes — DPOCK Macro _____ 6-34

Generating Area Clearance
Tool Paths — AREAMILL _____ 6-35
Setting Defaults for Area Clearance — DEFAMILL _____ 6-36
Defining Boundaries and Islands — AREAMILL _____ 6-37
Performing Area Clearance Operations _____ 6-38
Initial Operation _____ 6-39
Initial and Final Contouring Passes _____ 6-40
Lace Cutting _____ 6-40
Positioning _____ 6-41
Adding Machine Control Statements _____ 6-43
Using AREAMILL Point Macros _____ 6-44
Generating Output with AREAMILL Point Macros _____ 6-45
Using Variables _____ 6-45
Example of an AREAMILL Point Macro _____ 6-46

Tool Motion Generation: Hole Processing

Overview of Hole Processing Commands _____ 7-2

Hole Processing Methods _____ 7-2
Method 1 _____ 7-3
Method 2 _____ 7-3
Displaying Tool Motion _____ 7-3
Drilling — DRILL _____ 7-4
Specifying Drilling Methods _____ 7-5
Boring — BORE _____ 7-9
Specifying Boring Methods _____ 7-10
Countersinking and Chamfering — CSINK _____ 7-12
Tapping — TAP _____ 7-13
Controlling Hole Depth _____ 7-15
Controlling Hole Depth During a BORE, DRILL, or TAP _____ 7-15
Controlling Hole Depth During a CSINK _____ 7-18
Controlling Clearance Distances _____ 7-19
Identifying Locations and Order of Machining _____ 7-21
Setting Dwell Time _____ 7-22
Setting Avoidance Parameters _____ 7-23
Hole Processing on a Cylindrical Part _____ 7-24
Displaying Machine Tool Motion _____ 7-26

Tool Motion Generation: Noncutting

Overview of Noncutting Commands _____ 8-2
Positioning for a New Cut — APPROACH _____ 8-3
Withdrawing the Tool — RETRACT _____ 8-4
Moving to a Clearance Position — CLEAR _____ 8-5
Moving in Any Direction — MOVE _____ 8-6

Glossary

Preface

CVNC™-M2 User Guide provides instructions on all CVNC-M2 commands. It is written for NC programmers, operators, and manufacturing engineers using CVNC-M2 to produce control tapes for numerically controlled machine tools.

Related Documents

The following documents may be helpful as you use *CVNC™-M2 User Guide*:

- *Understanding CVNC*
- *CVNC System User Guide and Menu Reference*
- *CVNC Editor Guide*
- *Customizing CVNC*
- *CVNC Command and Interface Cross-Reference*
- *CVNC System Variables Guide*
- *CVNC Work Examples*
- *CVNC Master Index*

Book Conventions

The following table illustrates and explains conventions used in writing about CADDS applications.

Convention	Example	Explanation
Menu selections and options	List Section option, Specify Layer field	Indicates a selection you must make from a menu or property sheet or a text field that you must fill in.
User-selected graphic location	X, d_1 or P1	Marks a location or entity selection in graphic examples.
User input in CADDS text fields and on any command line	`cvaec.hd.data.param` `tar -xvf /dev/rst0`	Enter the text in a CADDS text field or on any command line.
System output	`Binary transfer complete.`	Indicates system responses in the CADDS text window or on any command line.
Variable in user input	`tar -cvf /dev/rst0` filename	Replace the variable with an appropriate substitute; for example, replace filename with an actual file name.
Variable in text	tagname	Indicates a variable that requires an appropriate substitute when used in a real operation; for example, replace tagname with an actual tag name.
CADDS commands and modifiers	INSERT LINE TANTO	Shows CADDS commands and modifiers as they appear in the command line interface.
Text string	`"SRFGROUPA"` or `'SRFGROUPA'`	Shows text strings. You must enclose text string with single or double quotation marks.
Integer	n	Supply an integer for the *n.*
Real number	x	Supply a real number for the *x*.
#	`# mkdir /cdrom`	Indicates the root (superuser) prompt on command lines.
%	`% rlogin` remote_system_name `-l root`	Indicates the C shell prompt on command lines.
$	`$ rlogin` remote_system_name `-l root`	Indicates the Bourne shell prompt on command lines.

Window Managers and the User Interface

According to the window manager that you use, the look and feel of the user interface in CADDS can change. Refer to the following table:

Look and Feel of User Interface Elements

User Interface Element	Common Desktop Environment (CDE) on Solaris, HP, and IBM	Window Manager Other Than CDE on Solaris, HP, IBM, and Windows
Option button	ON — Round, filled in the center OFF — Round, empty	ON — Diamond, filled OFF — Diamond, empty
Toggle key	ON — Square with a check mark OFF — Square, empty	ON — Square, filled OFF — Square, empty

Online User Documentation

Online documentation for each book is provided in HTML if the documentation CD-ROM is installed. You can view the online documentation in the following ways:

- From an HTML browser
- From the Information Access button on the CADDS desktop or the Local Data Manager (LDM)

Please note: The LDM is valid only for standalone CADDS.

You can also view the online documentation directly from the CD-ROM without installing it.

From an HTML Browser:

1. Navigate to the directory where the documents are installed. For example,

   `/usr/apl/cadds/data/html/htmldoc/` (UNIX)

   Drive:`\usr\apl\cadds\data\html\htmldoc\` (Windows)
2. Click `mainmenu.html`. A list of available CADDS documentation appears.
3. Click the book title you want to view.

From the Information Access Button on the CADDS Desktop or LDM:

1. Start CADDS.
2. Choose Information Access, the i button, in the top-left corner of the CADDS desktop or the LDM.
3. Choose DOCUMENTATION. A list of available CADDS documentation appears.
4. Click the book title you want to view.

From the Documentation CD-ROM:

1. Mount the documentation CD-ROM.
2. Point your browser to:

   CDROM_mount_point/htmldoc/mainmenu.html (UNIX)

   CDROM_Drive:\htmldoc\mainmenu.html (Windows)

Online Command Help

You can view the online command help directly from the CADDS desktop in the following ways:

- From the Information Access button on the CADDS desktop or the LDM
- From the command line

From the Information Access Button on the CADDS Desktop or LDM:

1. Start CADDS.
2. Choose Information Access, the i button, in the top-left corner of the CADDS desktop or the LDM.
3. Choose COMMAND HELP. The Command Help property sheet opens displaying a list of verb-noun combinations of commands.

From the Command Line: Type the exclamation mark (!) to display online documentation before typing the verb-noun combination as follows:

```
#01#!INSERT LINE
```

Printing Documentation

A PDF (Portable Document Format) file is included on the CD-ROM for each online book. See the first page of each online book for the document number referenced in the PDF file name. Check with your system administrator if you need more information.

You must have Acrobat Reader installed to view and print PDF files.

The default documentation directories are:

- /usr/apl/cadds/data/html/pdf/doc_number.pdf (UNIX)
- CDROM_Drive:\usr\apl\cadds\data\html\pdf\doc_number.pdf (Windows)

Resources and Services

For resources and services to help you with PTC (Parametric Technology Corporation) software products, see the *PTC Customer Service Guide*. It includes instructions for using the World Wide Web or fax transmissions for customer support.

Documentation Comments

PTC welcomes your suggestions and comments. You can send feedback electronically to `doc-webhelp@ptc.com`.

Chapter 1

Overview of CVNC-M2

This chapter provides an overview of CVNC-M2, a language-based, interactive, graphic software package that generates part programs for milling and hole processing operations.

- Purpose of CVNC-M2
- CVNC-M2 Features
- How to Enter CVNC-M2
- What CVNC-M2 Commands Do
- How to Generate and Edit a Job Control File

Purpose of CVNC-M2

CVNC-M2 generates part programs for milling and hole processing operations. When these part programs are processed, they produce control tapes for 2- and 2-1/2-axis NC (numerical control) milling machines and machining centers.

CVNC-M2 is a language-based, interactive, graphic software package. Using CVNC's command-driven language, you can control the sequence of events that define, set up, and execute a job containing a set of milling and/or hole processing operations. This sequence of events is called an NC process.

With CVNC-M2, you can specify point-to-point and continuous-motion tool paths, including circular interpolation. You can also specify tool and table rotations to generate tool paths in any orientation.

CVNC-M2 generates tool motion relative to part models constructed with any of the following CADDS entities:

- Points
- Lines
- Arcs
- Circles
- Ellipses
- Conics
- Parabolics
- Splines, B-splines, and Nsplines

Tasks Performed by CVNC-M2

With CVNC-M2, you can create part programs that perform the following NC functions:

- Defining and selecting tools
- Specifying milling parameters
- Point-to-point machining
- Profile milling
- Pocket milling
- Lace cutting

- Face milling
- Postprocessing operations

Instructions for using CVNC-M2 commands to perform these functions appear in this book.

CVNC-M2 Features

CVNC-M2 software uses features within the CVNC environment that give you flexibility and control over jobs.

The software package contains a default language environment comprising CVNC base system commands, NC commands for 2- and 2 1/2-axis milling and hole processing, a set of output pass-through statements, and statements to invoke CVMAC macros. CVMAC is a proprietary programming language.

You can modify the default language to add your own job-specific macros and output pass-through statements. For information about creating macros, see *Customizing CVNC.*

Pass-through statements, which are postprocessor commands included in the CVNC grammar files, can be entered like CVNC commands in your part programs. CVNC does not process them. Instead, they pass through to the APT, CLfile, or COMPACT II output file. For details on output generation, see the *CVNC System User Guide and Menu Reference*.

Job Control Files

CVNC-M2 stores the NC process commands in a text file called a Job Control File (JCF). CVNC automatically generates a JCF when you enter the milling module and begin executing CVNC commands. A typical JCF contains NC commands, base system commands, and output pass-through statements.

The JCF is a permanent record of your part program that you can reexecute or edit as needed. CVNC offers a set of editing commands for modifying and manipulating data in the JCF. For more information, see the *CVNC Editor Guide*.

This book focuses on how to use CVNC-M2 NC commands and macros. For information on CVNC base system and editing commands, refer to the *CVNC System User Guide and Menu Reference* and *CVNC Editor Guide*.

System Variables

CVNC-M2 maintains system variables that contain machining, system, and user-defined parameters resulting from the execution of CVNC commands and macros in the JCF. These parameters are always accurate with respect to the currently active line in the JCF.

You can reference system variables to find current values such as the active feed and speed rates or the radius or name of the active tool. See the *CVNC Master Index* for a complete listing of all system variables, and the *CVNC System User Guide and Menu Reference* for details on using system variables.

System Monitoring Tools

You can examine the current value of a system variable at any time. The value can be textual or numeric. You can also set up CADDS status windows to continually display and update the variables you want to see.

For more details, see the SHOW, PRINT, ASSIGN, and STATUS commands in the *CVNC System User Guide and Menu Reference.*

Macros

With macros, you can include logic programming, such as control, branching, and looping functions, in your JCF.

You can customize and enhance your interface and programming environment by writing your own macros in CVMAC.

For more information on building macros, refer to *Customizing CVNC*.

Command Files

Command files are text files containing a sequence of NC commands that perform repetitive functions, such as moving to the home point or changing the tool.

Command files contain no logic. You can write them as standalone executable files and call them as needed during any NC process.

For more information, refer to the *CVNC Editor Guide*.

Output Formats

CVNC-M2 converts the data in the JCF to one of three output formats supported by CVNC:

- APT source files
- CLfile binary formats

- COMPACT II source files

Specify the output you want with the OUTPUT command (described in the *CVNC System User Guide and Menu Reference*). APT and COMPACT II must be processed offline. CLFiles may be postprocessed online or offline.

The output process is controlled by an output generator, which normally executes in the background and is separate from the CVNC work session. This allows you to use CADDS as soon as you leave a CVNC work session.

Output Pass-through Statements

CVNC-M2 includes many output pass-through statements modified from the APT, CLfile, and COMPACT II languages. CVNC pass-through statements used with the CVNC-M2 package are described in the *CVNC System User Guide and Menu Reference*.

Pass-through statements do not affect CVNC processing, but they are sometimes needed to control output file processing or the machine tool. They can be used like CVNC commands in a JCF. CVNC-M2 passes these statements directly to the output file.

You can add pass-through statements to the system by using CVNC's ncgram append utility. For more information, see *Customizing CVNC*.

How to Enter CVNC-M2

Enter the CVNC milling module by typing the CADDS command PROGram NCmill. You can perform NC milling operations only on CADDS parts.

NC milling operations can be performed only on parts designed with CADDS entities, and you can only enter CVNC from CADDS.

The *CVNC System User Guide and Menu Reference* explains how to manually or automatically tag all currently untagged CADDS entities or to reject commands that refer to untagged entities.

To enter the milling module in CVNC, follow the instructions in the *CVNC System User Guide and Menu Reference.*

Once you are in the milling module, CVNC displays the application-level prompt, `NC:>.` You can now begin to develop your machining operations by using the relevant CVNC commands.

What CVNC-M2 Commands Do

CVNC-M2 commands direct milling or hole processing operations and control the job and programming processes. You will use CVNC base system commands and CVNC-M2 commands to create your milling and hole processing programs.

Base System Commands

Base CVNC commands include system commands and editing commands. They allow you to do the following:

- Control system graphics (for example, DISPLAY and GRAPHIC).
- Move between CADDS and CVNC as required (~, RESU, and IC).
- Monitor system variables (ASSIGN, STATUS, PRINT, and SHOW).
- Call and execute CVMAC macros and command files.
- Decompose NC commands for internal analysis and modification (IN and INX).
- Move backward and forward in your JCF (B and F editor commands).
- Edit text in the JCF.
- Add and modify CADDS part geometry while you are in the CVNC environment.
- Prevent data loss (SECURE).

Base system CVNC commands are described in detail in the *CVNC System User Guide and Menu Reference and CVNC Editor Guide*.

CVNC-M2 Commands

CVNC-M2 commands and modifiers direct one or more tools through a series of milling and/or hole processing operations. Base system and setup commands are included to control and individualize each job.

CVNC-M2 commands let you determine

- Job setup
- Operation setup
- Motion

This book focuses on how to use CVNC-M2 commands and macros, which are listed by function.

Job Setup Commands

Job setup commands allow you to specify the constant parameters required for all aspects of your milling and/or hole processing job. These parameters include configuration of the machining setup and characteristics of the tools used for cutting operations.

You should execute job setup commands before any other commands in the JCF.

Job setup commands are as follows:

Table 1-1 Machine Configuration

CONFIG	HOMEPT
DATUM	

Table 1-2 Tool Definition

DEFTOOL	LISTOOL
DELTOOL	TLIB

Operation Setup Commands

Operation setup commands allow you to specify parameters regulating individual cutting operations within a JCF.

These commands load the tool you want to use, select the construction plane you want to work in, and set operational parameters such as feed rates, speeds, and z-planes.

Operation setup commands are as follows:

Table 1-3 Regulating Cutting Operations

CHGTOOL	INDEX
CPL	TLCHG

Table 1-4 Operational Parameters

CALCRAD	PLANE
COOLANT	SPEED
DIACOMP	STOCK
FEED	TOLER

Motion Commands

The two main cutting actions of CVNC-M2 are milling and hole processing. Motion commands also include noncutting tool movements.

Motion commands are as follows:

Table 1-5 Milling

AREAMILL	PLUNGE
CUT	POCKET
DEFAMILL	PROFILE
DPOCK	SPROF
DPROF	

Table 1-6 Hole Processing

BORE	DRILL
CSINK	TAP

Table 1-7 Noncutting

APPROACH	MOVE
CLEAR	RETRACT

How to Generate and Edit a Job Control File

A Job Control File (JCF) is generated automatically when you begin executing CVNC-M2 commands. You can edit your JCF with editing commands.

Generating a JCF

After you enter the milling module and receive the `NC:>` prompt, generate a JCF by entering CVNC-M2 commands.

Please note: If you are using the CVNC icons and property sheets to execute the application, refer to the *CVNC System User Guide and Menu Reference* for instructions.

Enter commands using the keyboard or the mouse. The input sequence is generally command-modifier-value.

Some modifiers require no value; others require several values. Some modifiers require entity or location selections. CVNC checks syntax word-by-word and traps typographical errors immediately.

CVNC-M2 processes the command immediately, updating appropriate system variables and graphics.

Editing Data in a JCF

The *CVNC Editor Guide* provides instructions for editing a JCF.

Chapter 2

Job Setup: Machine Configuration

This chapter describes the DATUM, HOMEPT, and CONFIG commands that are used at the beginning of your Job Control File. The CONFIG command is only used for programming 4-axis or 5-axis milling machines.

- Initial Setup: Overview
- Defining the Machine Tool Linear Coordinate System and Part Program Zero — DATUM
- Specifying Home Location — HOMEPT
- Defining Milling Machines with Rotary Axes — CONFIG

Initial Setup: Overview

At the beginning of your JCF, use the DATUM, HOMEPT, and (if programming a 4- or 5-axis milling machine) CONFIG commands.

To set up your job, follow this sequence:

1. Use the DATUM command to identify a construction plane (Cplane), which defines the machine tool coordinate system xyz. All data points in the output file are given with respect to the coordinate system of the DATUM Cplane.
2. Use the HOMEPT command to define the initial starting point of the machine head.
3. If you are using any rotary axes machine tools, use the CONFIG command to define the axes.

Please note: If you do not have 4- or 5-axis machine tools, do not use the CONFIG command. You can define 4- or 5-axis machines in CVNC-M2 and CVNC-M3 that can be used for positioning.

Use the DATUM, HOMEPT, and CONFIG commands in this order, because each references information supplied in the previous one. For example, HOMEPT is defined with respect to DATUM. Rotary axes are referred to in CONFIG with respect to the machine tool x-, y-, and z-axes defined in DATUM.

The DATUM, HOMEPT, and CONFIG commands described in this chapter are used in all CVNC milling products.

Defining the Machine Tool Linear Coordinate System and Part Program Zero — DATUM

The DATUM command defines part program zero and the linear coordinate system of the machine tool (XYZ) in relation to the CADDS part.

Use the DATUM command to define the machine's linear (xyz) coordinate system orientation with respect to the CADDS model. The cutting tool axis is considered to be parallel to the z-axis of the machine tool.

DATUM remains constant. After you set DATUM, you can then select any active Cplane, Chapter 4, "Overview of Regulating Cutting Operations" as long as you do not change the z-axis.

DATUM sets the basic machine-tool relationship and the active construction plane.

CVNC works with relation to active construction space (a Cplane), but the output is always mapped to DATUM.

DATUM uses a Cplane defined in CADDS. The origin of the DATUM Cplane specifies the NC program x0 y0 z0 (output coordinate system).

For example, to use the Cplane named TOP as the DATUM, enter

```
NC:> DATUM "TOP"
```

Enclose the Cplane name in single or double quotation marks.

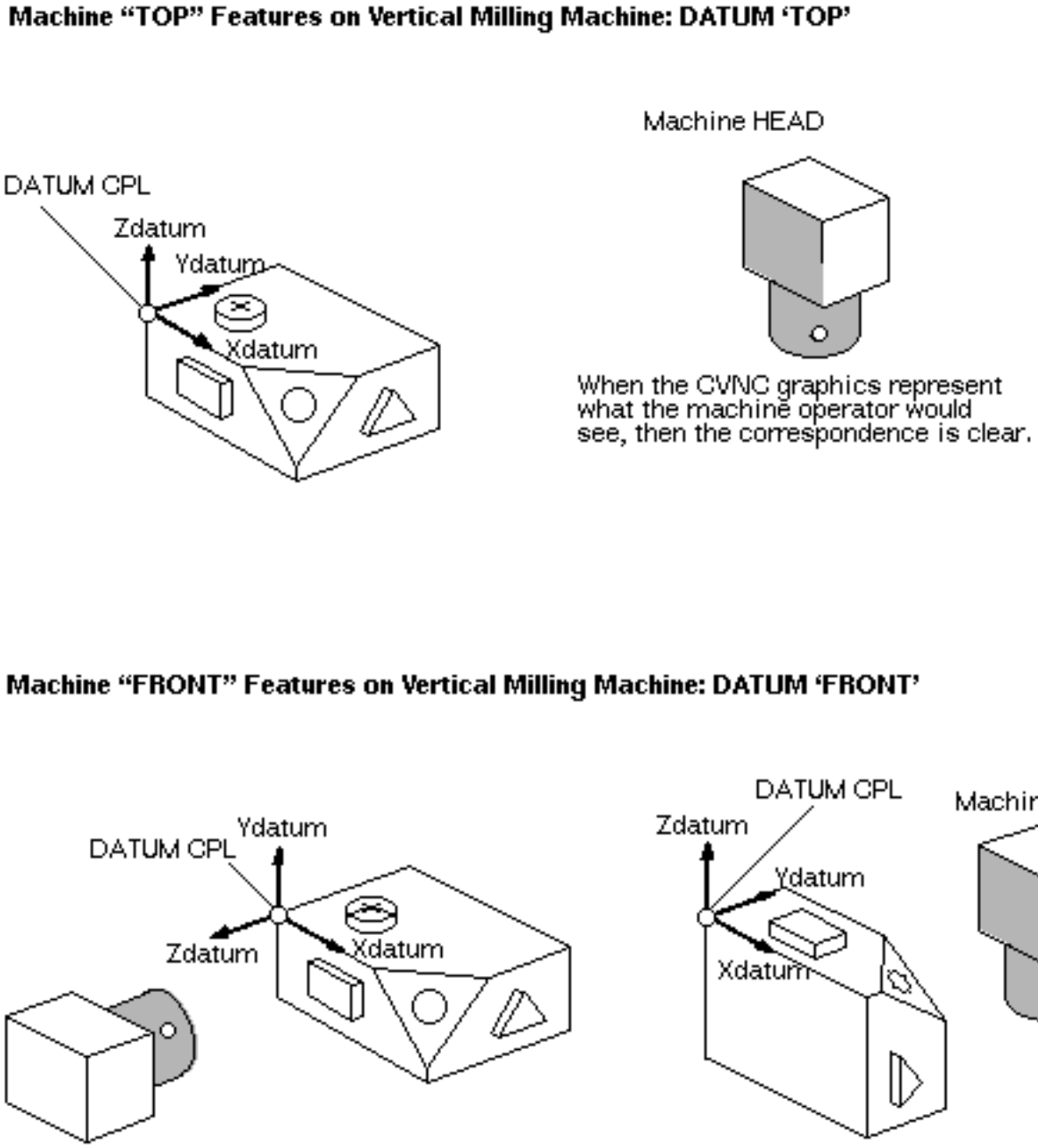
All data points in the output file are given with respect to the coordinate system of the DATUM Cplane.

Please note:

1. The graphic representation in CVNC shows the machine "wrapped" around the CADDS model; the model is static. However, in actual use, you would be moving the model to various positions.
2. While using a Cplane as a DATUM Cplane in a JCF, do not delete or redefine that Cplane outside CVNC. In case you have deleted or redefined the Cplane outside CVNC, execute and save the JCF that uses this Cplane as a DATUM Cplane.

Examples of these two views are shown in the illustration given below.

Figure 2-1 Wrapping the Machine Around the CADDS Model


Machine "TOP" Features on Vertical Milling Machine: DATUM 'TOP'
DATUM CPL
Zdatum
Ydatum
Xdatum
Machine HEAD
When the CVNC graphics represent what the machine operator would see, then the correspondence is clear.
Machine "FRONT" Features on Vertical Milling Machine: DATUM 'FRONT'
Ydatum
DATUM CPL
Zdatum
Xdatum
DATUM CPL
Zdatum
Ydatum
Xdatum
Machine HEAD
CADDS Representation
Based on wrapping machine around the part.
In actual practice you would, instead, move the part.
Actual Machine Tool Orientation
The way it would look in actual use: The part/material is placed into machine, not machine around part.

Specifying Home Location — HOMEPT

The HOMEPT command defines the starting point and home location for the machine head (gage reference line or spindle nose).

Use the HOMEPT command to define the initial position of the machine tool head (the gage reference line or spindle nose). This position is defined as an xyz coordinate with respect to the DATUM origin. See the figure on the following page.

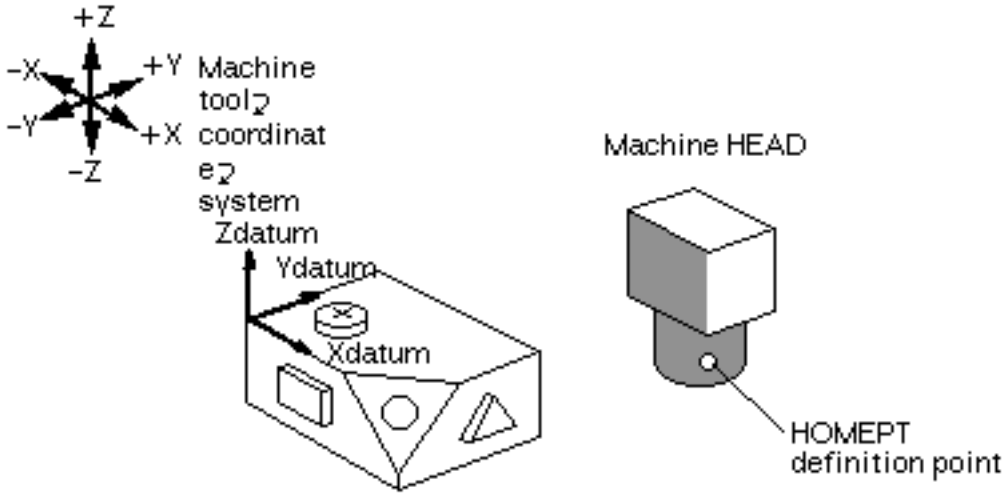
HOMEPT FROM (the HOMEPT command with the FROM modifier) specifies a starting and home location for the machine spindle and tool. This location represents the tool change location.

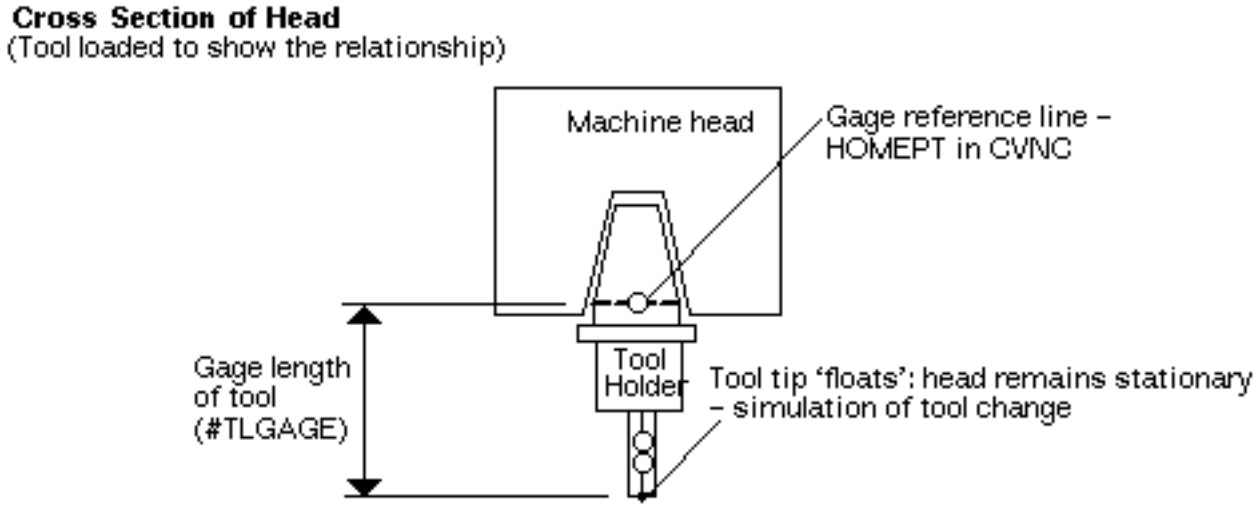
Enter the HOMEPT command after the DATUM command and before any CHGTOOL or CONFIG commands.

HOMEPT has the following characteristics and requirements:

- If not specified, #HOMEPT (the HOMEPT variable) defaults to x 0, y 0, z 20.0 inches (or 500 mm).
- HOMEPT should accommodate the maximum tool gage length.
- HOMEPT remains constant, with respect to the machine, regardless of rotary device motion.
- You must specify HOMEPT FROM before using the CONFIG command. This is essential for machines containing rotary head devices.
- You must use HOMEPT FROM to output a FROM statement to your output file.

Figure 2-2 HOMEPT of the Machine Head

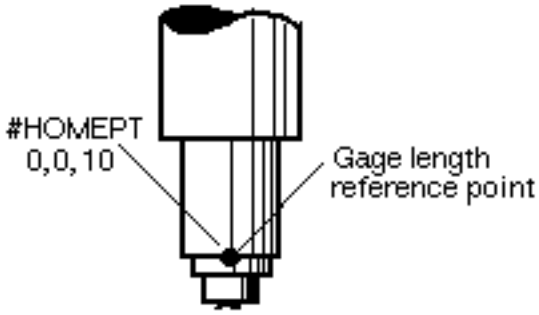



Cross Section of Head
(Tool loaded to show the relationship)
Machine head
Gage reference line – HOMEPT in CVNC
Gage length of tool (#TLGAGE)
Tool Holder
Tool tip 'floats': head remains stationary – simulation of tool change

Using HOMEPT Before a Tool Change

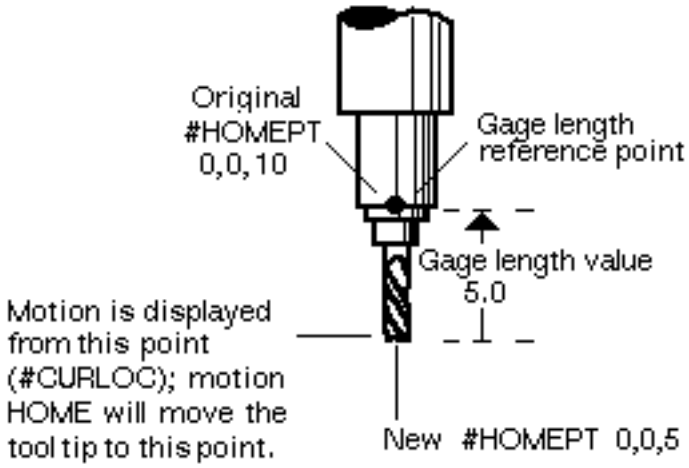
Specify HOMEPT FROM before a tool change by establishing HOMEPT from the gage length reference point of the spindle.

```
NC:> DATUM "CPLNAME"
NC:> HOMEPT FROM X 0 Y 0 Z 10 ;
```



When you execute the CHGTOOL command to activate a tool with a gage length greater than zero, tool motion is displayed from the new tool tip location (#CURLOC). CVNC adjusts the #HOMEPT and #CURLOC variables for the new tool gage length. Follow the example below to produce these results.

```
NC:> HOMEPT FROM X 0 Y 0 Z 10 ;
NC:> DEFTOOL "MILL 5.0" MILL7 DIA 2.0 CORNER 0.0 HEIGHT 3.0 GAGE
5.0
NC:> CHGTOOL 1 "MILL 5.0"
```



Defining Milling Machines with Rotary Axes — CONFIG

If you have 4- and 5-axis milling machines, use the CONFIG command to define the rotary axis capabilities.

The CONFIG command defines the rotary axis capability of the machine tool. If you do not have 4- or 5-axis machine tools, do not use CONFIG.

Use CONFIG to define each rotary axis in terms of

- The device type
  - Head: A rotary axis supported in the machine's head

    In this device, the cutting tool moves in the direction indicated by the CVNC output.
  - Table: A rotary axis supported in the machine's table

    In this device, the table moves in a direction opposite to that indicated by the CVNC output.
- The type of axis
  - A-axis: Rotation around the machine tool x-axis
  - B-axis: Rotation around the machine tool y-axis
  - C-axis: Rotation around the machine tool z-axis
- The position of the pivot axis at the time of setup, prior to machining
- The travel limits
  - Maximum: Value of the maximum travel limit
  - Minimum: Value of the minimum travel limit
  - Mininc: Value of the minimum increment

Please note: You can use CONFIG to define travel limits for the linear x-, y-, and z-axes.

Use CONFIG to describe your machine in relation to your part.

If one rotary axis is defined, the result is a 4-axis machine; two rotary axes result in a 5-axis machine. CONFIG can be used for both 4- and 5-axis machines, but for 5-axis machines the relationship between the two rotary axes must be defined.

You can define 4- and 5-axis milling machines within any CVNC milling application: CVNC-M2, CVNC-M3, and CVNC-M5. The use of rotary devices is restricted to positioning within CVNC-M2 and CVNC-M3 only. CVNC-M5 provides 4- and 5-axis surface machining and surface intersection.

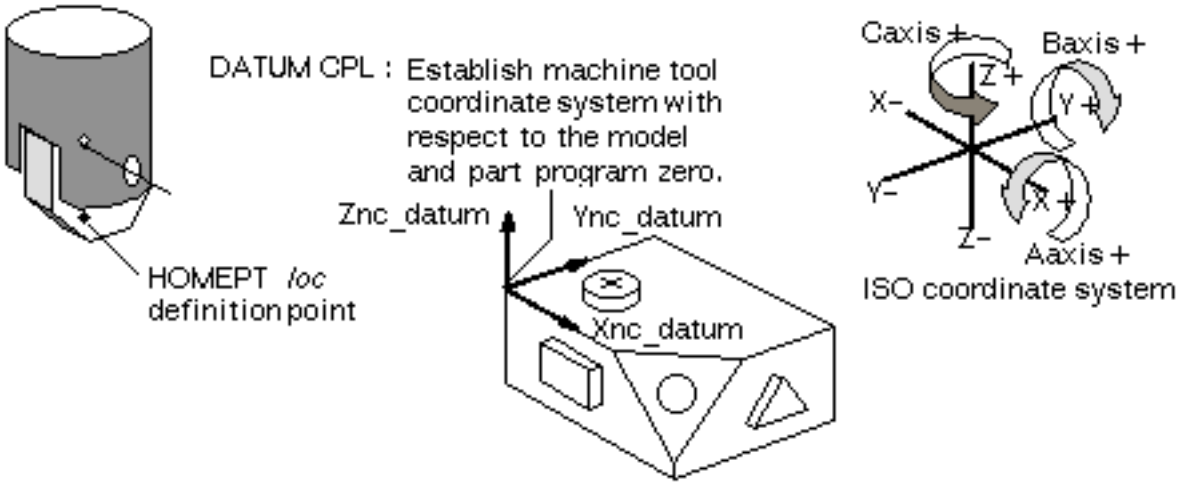
The distance between the gage reference point (HOMEPT) and the axis of rotation for a head is fixed by the machine tool builder. The difference is derived from the difference between the HOMEPT coordinate and the location of the rotary axis defined by CONFIG. (See also "Configuring 4-Axis Milling Machines" later in this chapter.)

The Relationship of the DATUM Cplane to Rotary Axes

You must execute DATUM prior to defining rotary axes with CONFIG to establish the linear coordinate system of the machine tool with respect to the model. Rotary axis definition is with respect to the DATUM Cplane.

The figure below illustrates these relationships.

Figure 2-3 Relationship of Rotary to Linear Axes; role of DATUM




```
NC:> DATUM nc datum
NC:> HOMEPT FROM X-100 Y-250 Z300 ;
```

Any rotary device is assumed to be in its zero position when defined; hence the tool axis aligns with the ZAXIS of DATUM.

Configuring Linear Axes: Overview

The CONFIG command supports rotary and linear travel limits. This means that you can define the range of motion of each axis of the target machine tool. All requested motions are then checked with these limits.

Defining the Travel Limits for Linear Motion

Use the CONFIG modifiers MAXIMUM and MINIMUM to specify the maximum and minimum values, respectively, of the coordinates for the specified linear axis.

Define a linear axis in the following format:

```
NC:> CONFIG XAXIS MAXIMUM exp MINIMUM exp
```

(In the above example you could specify `YAXIS` or `ZAXIS` instead of `XAXIS`.)

Validating Motion

All motions of the x-, y- and z-axes are checked to see that they are within the travel limits defined for that particular axis. If not within these limits, an error message is issued and the command terminates.

For example, the following sequence of commands issues an error:

```
NC:> CONFIG XAXIS MAXIMUM 1000 MINIMUM -1000
NC:> MOVE XYZLOC X1100 Y300 Z200
```

The resulting error message is:

```
NC:> ERROR:X coordinate position of 1100.0 requested.This
violates X MAXIMUM travel limit of 1000.0.Command terminated
with 1-lines of JCF pending.
```

Unsetting the Limits

If you do not specify any values for MAXIMUM and/or MINIMUM, they are unset to their default values, that is, +infinity and -infinity, respectively.

For example, the following command

```
NC:> CONFIG XAXIS MINIMUM
```

unsets the MINIMUM modifier for the x-axis to -infinity, and

```
NC:> CONFIG XAXIS
```

unsets the values of MAXIMUM and MINIMUM for the x-axis, simultaneously.

Configuring 4-Axis Milling Machines: Overview

Use the CONFIG modifiers AAXIS, BAXIS, and CAXIS to specify rotation around the x-, y-, and z-axes, respectively, for a machine with one rotary axis. Use the new modifiers MAXIMUM, MINIMUM and MININC to define the travel limits (in degrees) of each axis of the target machine tool.

Define a 4-axis milling machine in the following format.

For rotary head machines:

```
NC:> CONFIG AAXIS HEAD MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc
```

(In the above example you could specify `BAXIS` instead of `AAXIS`.)

For rotary table machines:

```
NC:> CONFIG AAXIS MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc
```

(In the above example you could specify `BAXIS` or `CAXIS` instead of `AAXIS`.)

Please note: If HEAD is not specified, CVNC assumes the rotary axis is in the table.

The rotary_axis_loc modifier is an xyz coordinate that defines the initial location of the rotary axis.

Examples of the CONFIG command for each specific machine are shown later in this chapter.

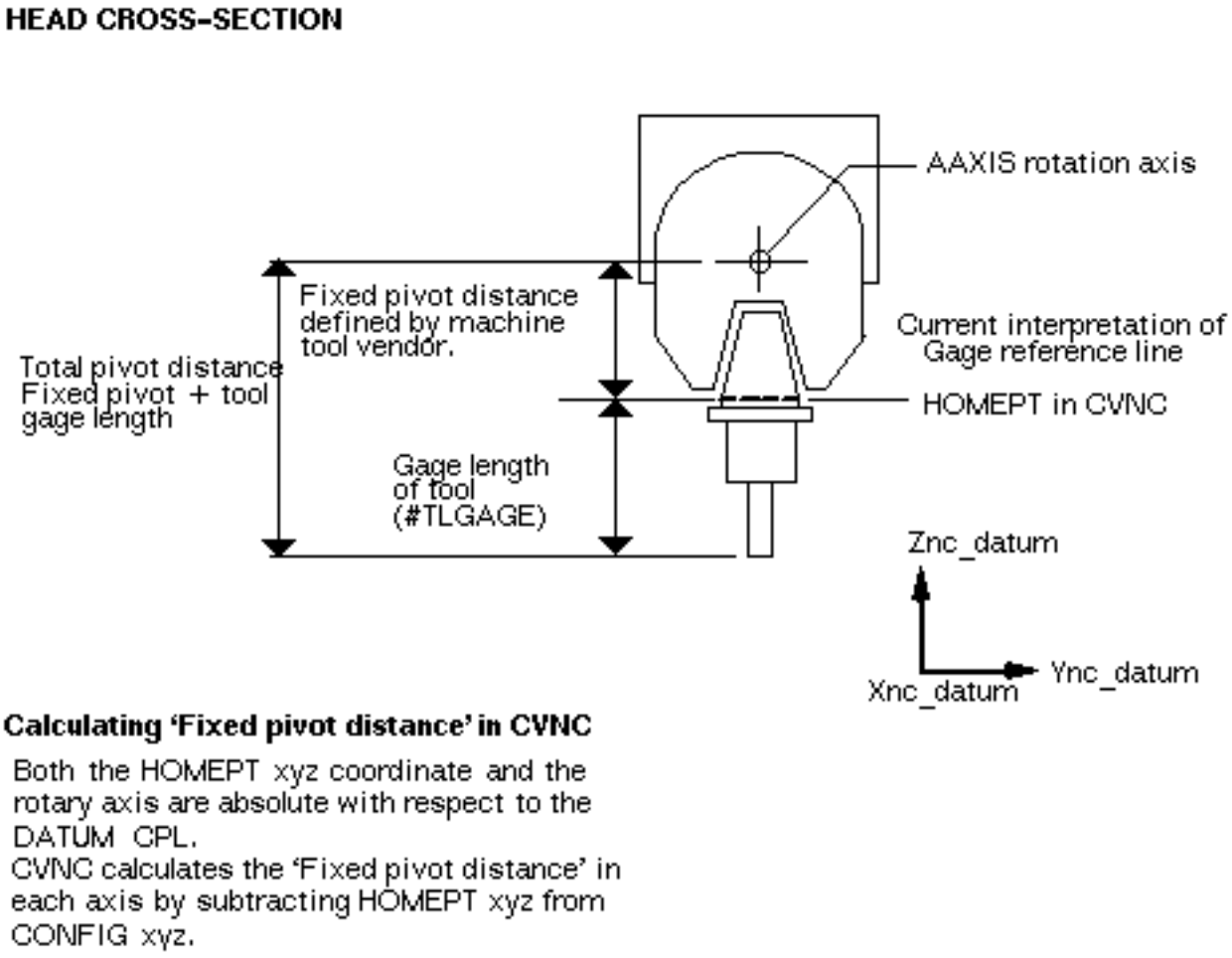
The Relationship of HOMEPT with CONFIG

Keep in mind the relationship of the HOMEPT position with respect to the location of the rotary axis defined in CONFIG, especially when defining a rotary head machine.

The figure on the following page shows how the relationship between the gage reference line and the rotary axis is determined from the locations specified.

The example here shows an AAXIS head machine, but the methodology is the same for any rotary head configuration.

Figure 2-4 Relationship Between Gage Reference Line and Rotary Axis


HEAD CROSS-SECTION
AAXIS rotation axis
Fixed pivot distance defined by machine tool vendor.
Current interpretation of Gage reference line
Total pivot distance Fixed pivot + tool gage length
HOMEPT in CVNC
Gage length of tool (#TLGAGE)
Znc_datum
Ync_datum
Xnc_datum
Calculating 'Fixed pivot distance' in CVNC
Both the HOMEPT xyz coordinate and the rotary axis are absolute with respect to the DATUM CPL.
CVNC calculates the 'Fixed pivot distance' in each axis by subtracting HOMEPT xyz from CONFIG xyz.

The relationship between the gage reference line and the rotary axis is fixed by the machine tool builder. When a tool assembly is loaded (CHGTOOL), its gage length (#TLGAGE) is added to the fixed pivot distance.

When you program the rotary device (see the INDEX command in Chapter 4), the whole assembly will rotate around the rotary axis. The tool path radius equals the total pivot distance. The position of the tool after an INDEX is thus directly affected by the relationships defined during the HOMEPT/CONFIG setup stage.

Defining the Travel Limits for Rotational Motion

You can define the range of motion (in degrees) of the rotary axes, that is, AAXIS, BAXIS and CAXIS.

- MAXIMUM `exp` defines the maximum value of the angle of movement
- MINIMUM `exp` defines the minimum value of the angle of movement
- MININC `exp` defines the minimum value of the step by which you can increase the angle of movement

All motions of the a-, b- and c-axes are checked to see that they are within the specified travel limits. If they are not so, an error message is issued and the command terminates.

For example, the following sequence of commands issues an error:

```
NC:> CONFIG AAXIS HEAD MAXIMUM 90 MINIMUM -90
NC:> INDEX AAXIS ATANGL -100
```

See “Programming Rotary Axes — INDEX” on page 4-8, for details.

The resulting error message is:

```
NC:> ERROR: Rotation -100.00 OF AAXIS IS OUTSIDE THE
ALLOWABLE RANGE MINIMUM -90.0 MAXIMUM 90.0 TYPED INPUT
TERMINATED
```

You can unset the modifiers to their default values in the same way as for the linear axes.

Please note: When you use the CONFIG and INDEX commands, CVNC generates output in the following format:

`ROTATE` axis `ATANGL` angle direction

where the angle is always an absolute value even if you specify negative maximum or minimum values for the angle of movement.

Resolving Multiple Solutions

When there is more than one way to position the machine tool, the travel limits may resolve the ambiguity.

For example, the following sequence of commands generates two possible solutions:

```
NC:> DATUM "TOP"
NC:> CONFIG CAXIS HEAD AAXIS HEAD MAXIMUM 90 MINIMUM 0 MININC 0.1
BOTH
NC:> INDEX "FRONT"
```

They are

1. CAXIS 0 AAXIS 90
2. CAXIS 180 AAXIS -90

In this case, the second solution extends beyond the limits of the a-axis and, so, the first solution is chosen. If both solutions are valid, the one that needs the least amount of movement is chosen.

Specifying the Direction of Rotation

You can specify the direction of rotation, that is, clockwise or anticlockwise. If you do not specify the direction, the one that does not extend beyond the travel limits is chosen. If positioning the machine tool violates the travel limits, an error message is issued and the command terminates.

For example, in the following sequence of commands the last INDEX command could either rotate clockwise or anticlockwise from the absolute position of BAXIS 270 to BAXIS 90. Both solutions are of equal distance and no direction is specified in the command. However, rotating anticlockwise causes the b-axis to pass through the 360 degree point, thus violating its limit. Hence, in this case, the clockwise solution is created.

```
NC:> DATUM "TOP"
NC:> CONFIG AAXIS MAXIMUM 90 MINIMUM 0 BAXIS MAXIMUM 360 MINIMUM
-360 BOTH
NC:> INDEX BAXIS ATANGL 180
NC:> INDEX BAXIS INCR 90
NC:> INDEX BAXIS ATANGL 90
```

An error message is issued if you specify the anticlockwise direction in the above example, as follows:

```
NC:> DATUM "TOP"
NC:> CONFIG AAXIS MAXIMUM 90 MINIMUM 0 BAXIS MAXIMUM 360 MINIMUM
-360 BOTH
NC:> INDEX BAXIS ATANGL 180
NC:> INDEX BAXIS INCR 90
NC:> INDEX BAXIS ATANGL 90 CCLW
```

Here, the anticlockwise direction specified in the last INDEX command makes the b-axis extend beyond its defined limits. An error message is issued and the command terminates.

Configuring a 4-Axis Milling Machine with a Rotary Head

Use CONFIG for a- or b-axis heads, preceded by the appropriate DATUM and HOMEPT commands.

With a 4-axis milling machine, you can use CONFIG to configure either an AAXIS head machine or a BAXIS head machine.

Configuring an A-Axis Head Machine

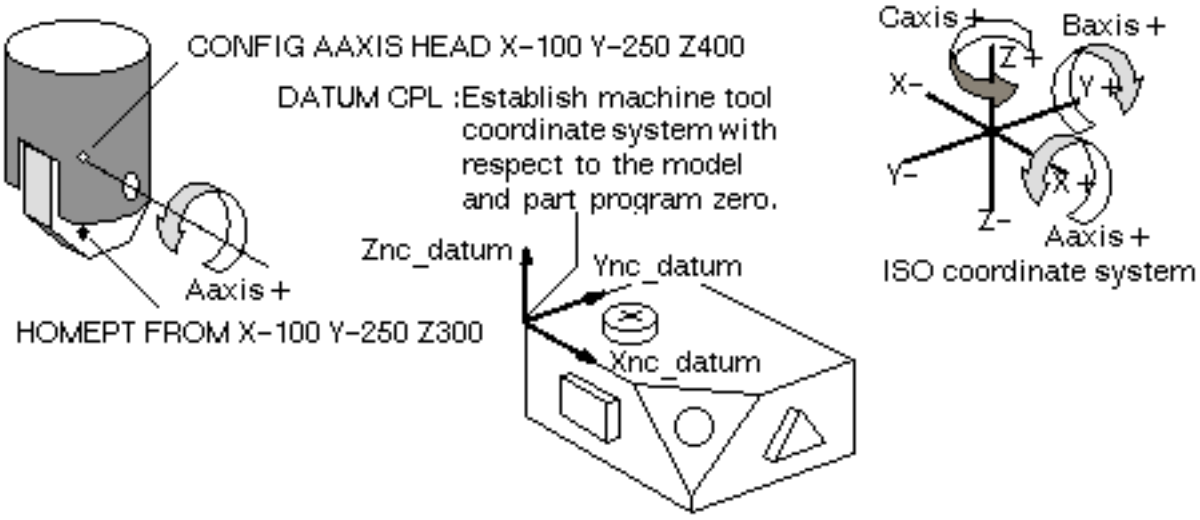
To configure an a-axis head machine, use the following command:

```
NC:> CONFIG AAXIS HEAD MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc
```

When the rotary axis is an a-axis, the rotation is around the x-axis of the machine tool.

The figure below shows the sequence of JCF commands and the graphical interpretation for definition of a 4-axis milling machine with an a-axis head. The fixed pivot distance is 100 (Z400 - Z300).

Figure 2-5 Machine Tool with A-Axis Head




```
NC:> DATUMnc datum
NC:> HOMEPT FROM X-100 Y-250 Z300 ;
NC:> CONFIG AAXIS HEAD X-100 Y-250 Z400
```

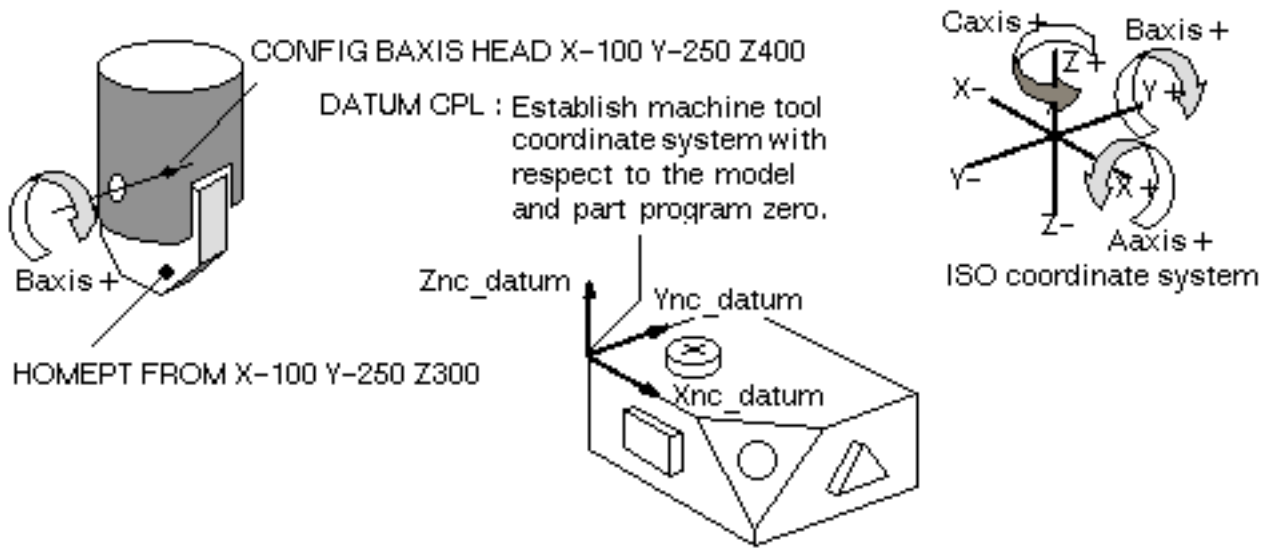
Configuring a B-Axis Head Machine

To configure a b-axis head machine, use the following command:

```
NC:> CONFIG BAXIS HEAD MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc
```

This is identical to the a-axis head example except that the rotation occurs around the y-axis of the machine tool. The figure shows the layout within CVNC.

Figure 2-6 Machine Tool with B-Axis Head



Please note: You cannot define a 4-axis machine tool with a rotary c-axis head in CVNC. Since the spindle rotates around the ZAXIS of the machine tool, definition of a c-axis rotary axis (acting around the z-axis) would be illogical.

Configuring a 4-Axis Milling Machine with a Rotary Table

You can configure a-, b-, or c-axis rotary tables. To view the part as you would actually machine it may aid your understanding of CVNC.

By far the most common 4-axis milling machine is a horizontal BAXIS table machine. CVNC supports the definition of a-, b-, and c-axis table configurations.

To configure a 4-axis table machine, use the following command format:

```
NC:> CONFIG AAXIS MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc
```

(In the above example you could specify `BAXIS` or `CAXIS` instead of `AAXIS`.)

CVNC assumes a rotary table; hence the omission of the word “HEAD” signifies a table. The rotary_axis_loc defines the position of the rotary axis with respect to the DATUM Cplane. This represents the action of placing the part on the table at a distance from the axis of rotation.

AAXIS Table Machine

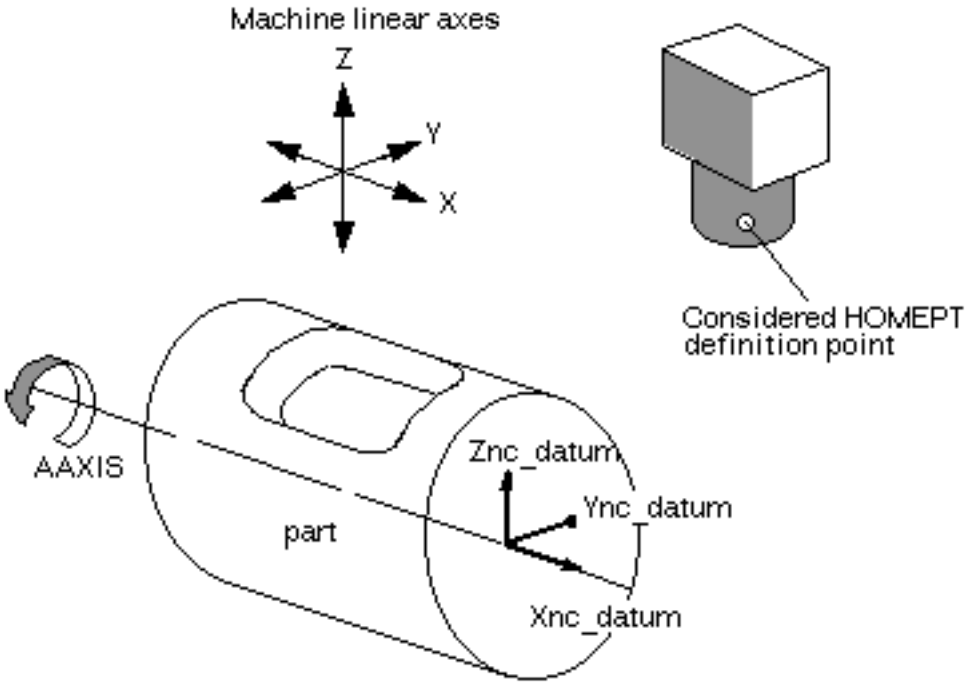
Use the CONFIG command as stated above to configure the a-axis table machine. For example,

```
NC:> CONFIG AAXIS X0 Y0 Z0 ;
```

AAXIS table machines are typically small, vertical machines with an add-on device that holds the part between centers.

The next figure shows the configuration of an a-axis table machine.

Figure 2-7 Rotary AAXIS Table on a Vertical Milling Machine



In above figure, the DATUM Cplane has an origin and x-axis coincident with the a-axis. The position of the AAXIS with respect to DATUM is therefore X0Y0Z0. HOMEPT defines the start point of the machine's head and is not directly associated with the rotary table.

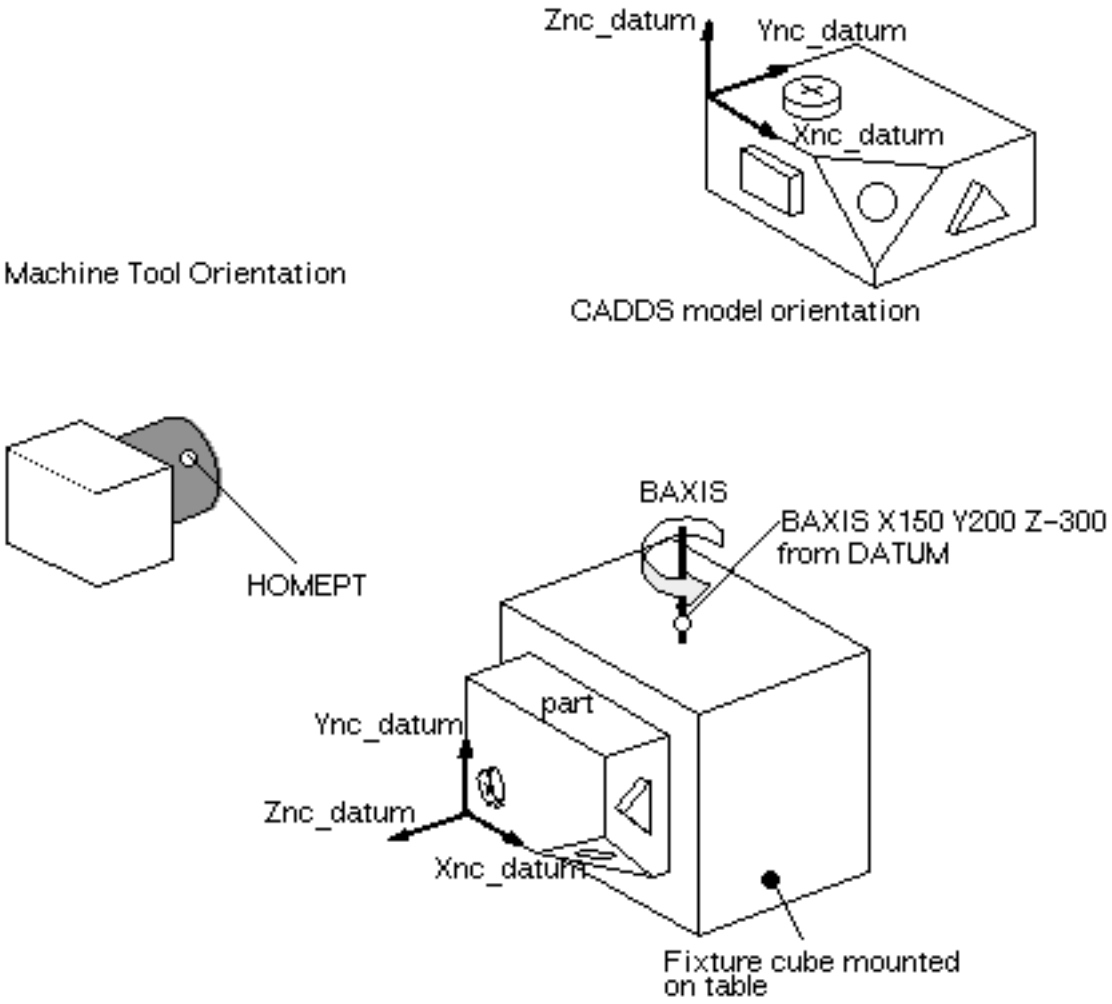
BAXIS Table Machine

Use the CONFIG command as stated above to configure the b-axis table machine. For example,

```
NC:> CONFIG BAXIS MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc
```

This is the most common 4-axis machine and is typically a horizontal machine. It is common for a fixture cube or angle plate to be mounted on the rotary table. The material from which the part is to be machined is located here, as shown in the following figure.

Figure 2-8 4-Axis Horizontal B-Axis Table Milling Machine (as it might appear in actual use)



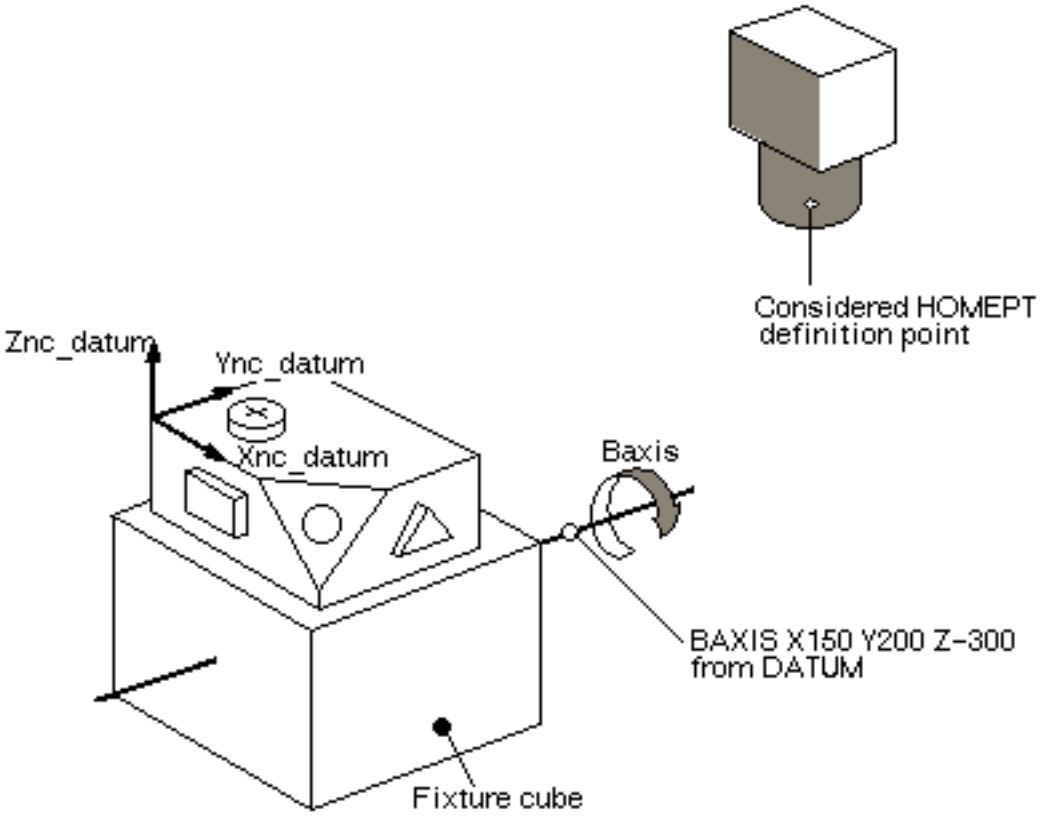

```
NC:> DATUM nc_datum
NC:> HOMEPT FROM X-100 Y200 Z500 ;
NC:> CONFIG BAXIS X150 Y200 Z-300 ;
```

The above figure is drawn from a machine tool orientation viewpoint and hence represents a view that would be seen from “behind” the machine if standing on the shop floor.

Bear in mind that CVNC views typically wrap the machine around the part (DATUM); thus the machine orientation may not correspond to the view you would see in actual use.

In the figure below, the model image (in CADDS, as shown in top corner) has been reoriented into the machine frame. The figure shows the result of wrapping the machine around the CADDS model.

Figure 2-9 4-Axis Horizontal B-Axis Table Milling Machine (CVNC Model-based View)




```
NC:> DATUM nc datum
NC:> HOMEPT FROM X-100 Y200 Z500 ;
NC:> CONFIG BAXIS X150 Y200 Z-300 ;
```

Mathematically, there is no difference between the above figure and the one on the previous page, but appreciation of this functionality will aid understanding when driving rotary axes. You may use this information when using the INDEX command.

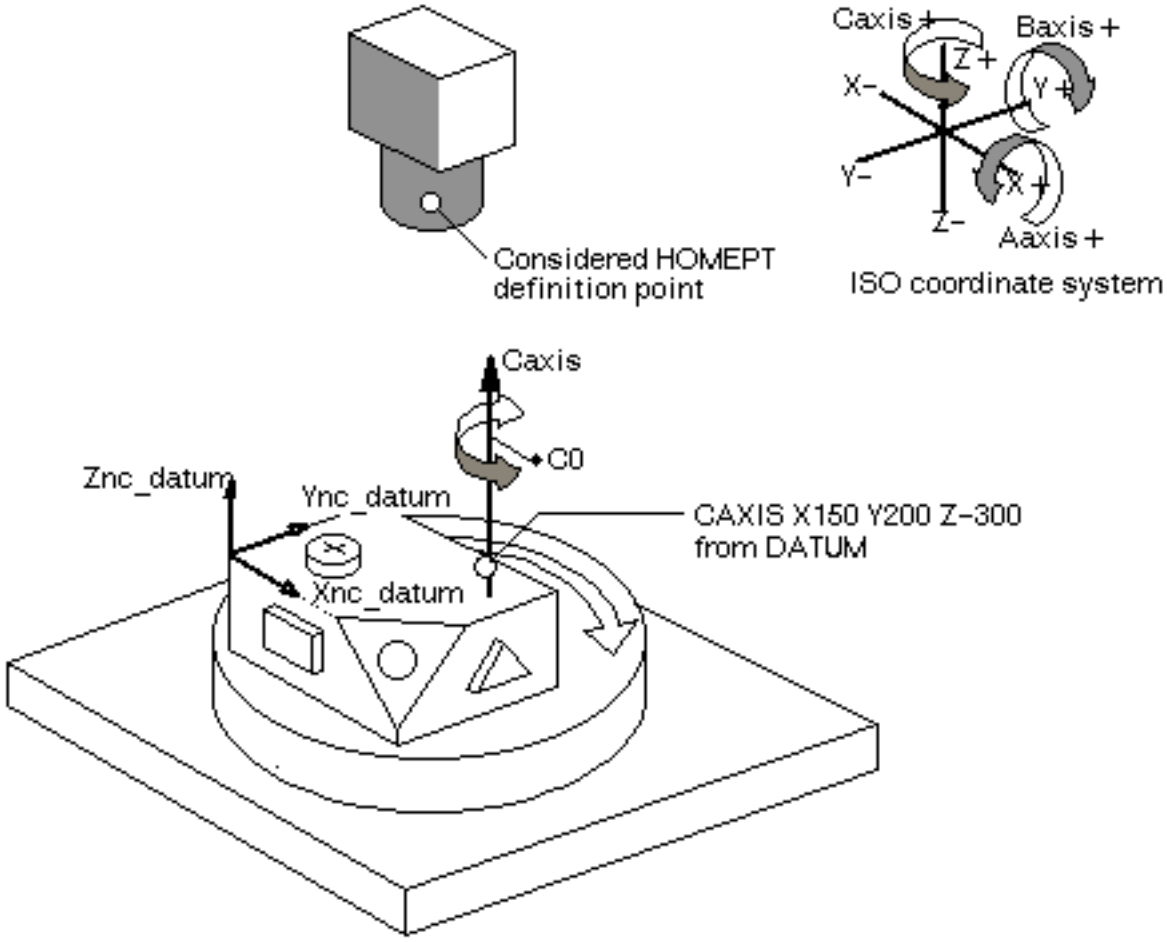
C-Axis Table Machine

Use the CONFIG command as stated above for this configuration:

```
NC:> CONFIG CAXIS MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc
```

The rotary table rotates around the machine's z-axis (in the xy-plane). This is shown in the figure below.

Figure 2-10 C-Axis Rotary Table




```
NC:> DATUM nc datum
NC:> HOMEPT FROM X-100 Y200 Z500 ;
NC:> CONFIG CAXIS X150 Y200 Z-300 ;
```

Configuring 5-Axis Milling Machines: Overview

CONFIG allows you to specify any of three combinations of rotary devices for 5-axis machines.

CVNC supports definition of the following 5-axis milling machine types:

- Compound head: Both rotary axes in the head
- Compound table: One rotary table mounted on top of another rotary table
- Head and table: one rotary axis in the head, one in the table

The term compound is used when both rotary axes are of the same type (both heads or both tables).

The order of definition is important, because it defines the relationship of the rotary devices.

Configuring 5-Axis Milling Machines with Compound Heads

Configure 5-axis milling machines with compound heads, specifying the axes of rotation.

You may configure 5-axis milling machines with compound heads that rotate around the CAXIS and AAXIS or around the CAXIS and BAXIS.

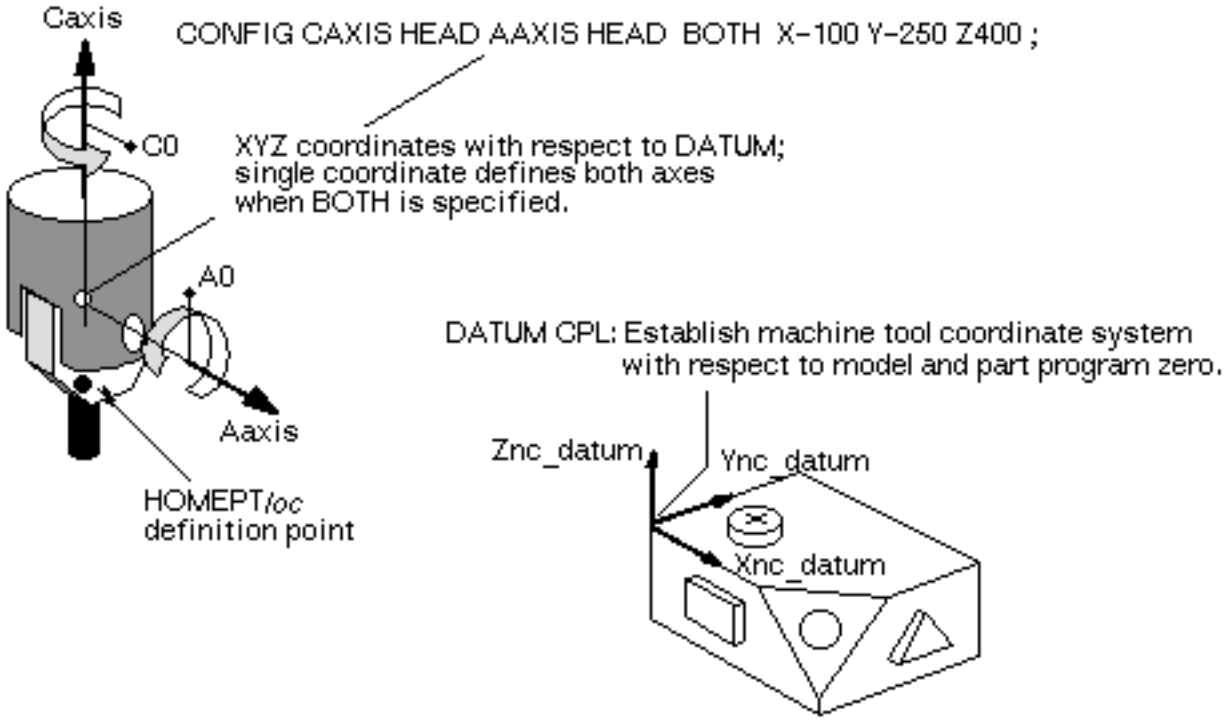
Compound Head: CAXIS and AAXIS Rotation

To configure a 5-axis milling machine with rotation around the CAXIS and AAXIS, use the following command:

```
NC:> CONFIG CAXIS HEAD MAXIMUM exp MINIMUM exp MININC exp AAXIS
HEAD MAXIMUM exp MINIMUM exp MININC exp BOTH loc
```

Here both rotary axes are mounted in the head and will typically be a vertical gantry type machine. An example of this configuration is shown in the next figure.

Figure 2-11 5-Axis Compound Head Milling Machine, C- and A-Axis Rotation




```
NC:> DATUM nc_datum
NC:> HOMEPT FROM X-100 Y-250 Z300 ;
NC:> CONFIG CAXIS HEAD AAXIS HEAD BOTH  X-100 Y-250 Z400 ;
NC:> CHGTOOL 1 '20MM S/DRILL'
```

AAXIS device is slaved to the CAXIS device since C rotation will cause the AAXIS device to rotate. If A rotates, the CAXIS device is unaffected.

Although BOTH is used in the command sequence in the above figure, the CONFIG command allows `loc` to be supplied for each rotary axis to define its starting location. For example, if the C and A axes in the figure were not coincident, you would use the following command:

```
NC:> CONFIG CAXIS HEAD MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc AAXIS HEAD MAXIMUM exp MINIMUM exp MININC exp
rotary_axis_loc
```

Notice the order of definition.

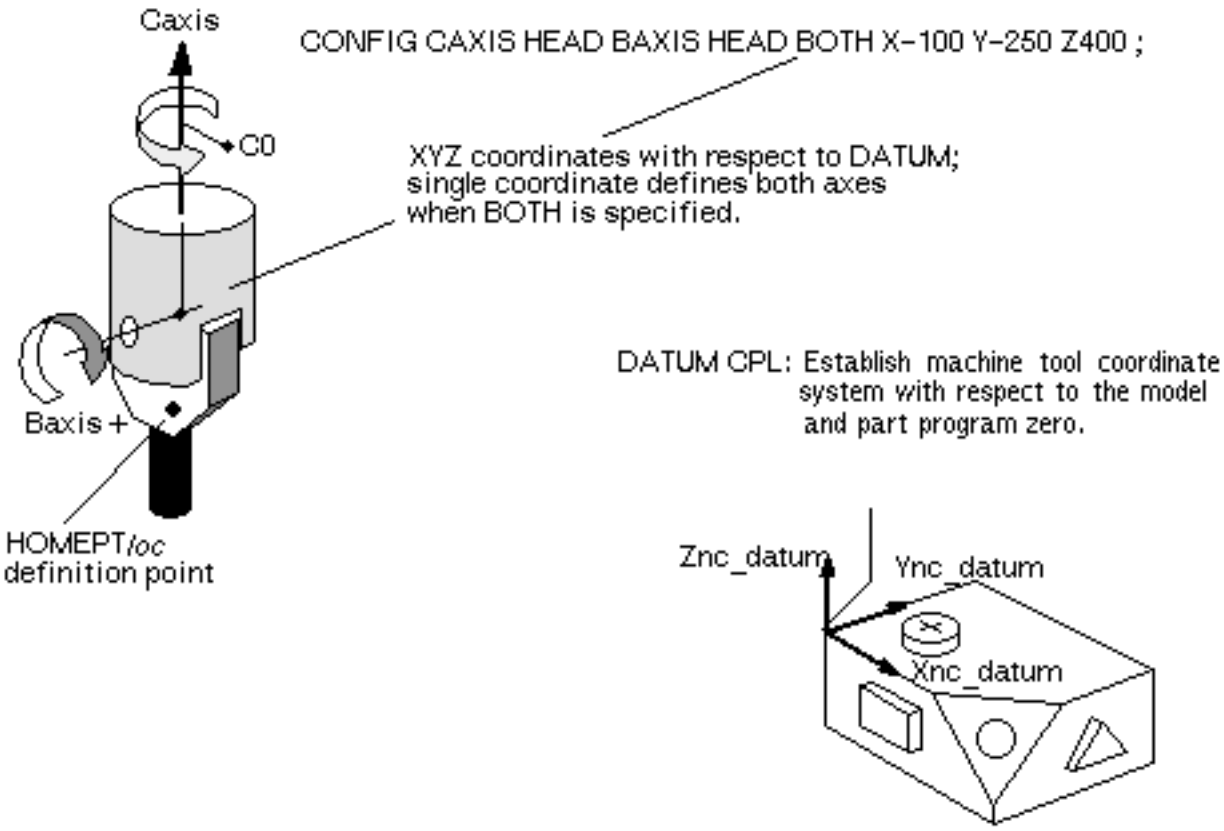
Compound Head: C- and B-Axis Rotation

For a 5-axis milling machine with rotation around the c- and b-axes, use the following command:

```
NC:> CONFIG CAXIS HEAD MAXIMUM exp MINIMUM exp MININC exp BAXIS
HEAD MAXIMUM exp MINIMUM exp MININC exp BOTH loc
```

Both rotary axes are mounted in the head and will typically be a vertical gantry type machine; the configuration is shown in the following figure.

Figure 2-12 5-Axis Compound Head Milling Machine, B-Axis Rotation




```
NC:> DATUM nc_datum
NC:> HOMEPT FROM X-100 Y-250 Z300 ;
NC:> CONFIG CAXIS HEAD BAXIS HEAD BOTH X-100 Y-250 Z400 ;
NC:> CHGTOOL 1 '20MM S/DRILL'
```

BAXIS device is slaved to the CAXIS device since C rotation will cause the BAXIS device to rotate. If B rotates, the CAXIS device is unaffected.

Configuring 5-Axis Milling Machines with Compound Tables

With CONFIG you can define a 5-axis compound table.

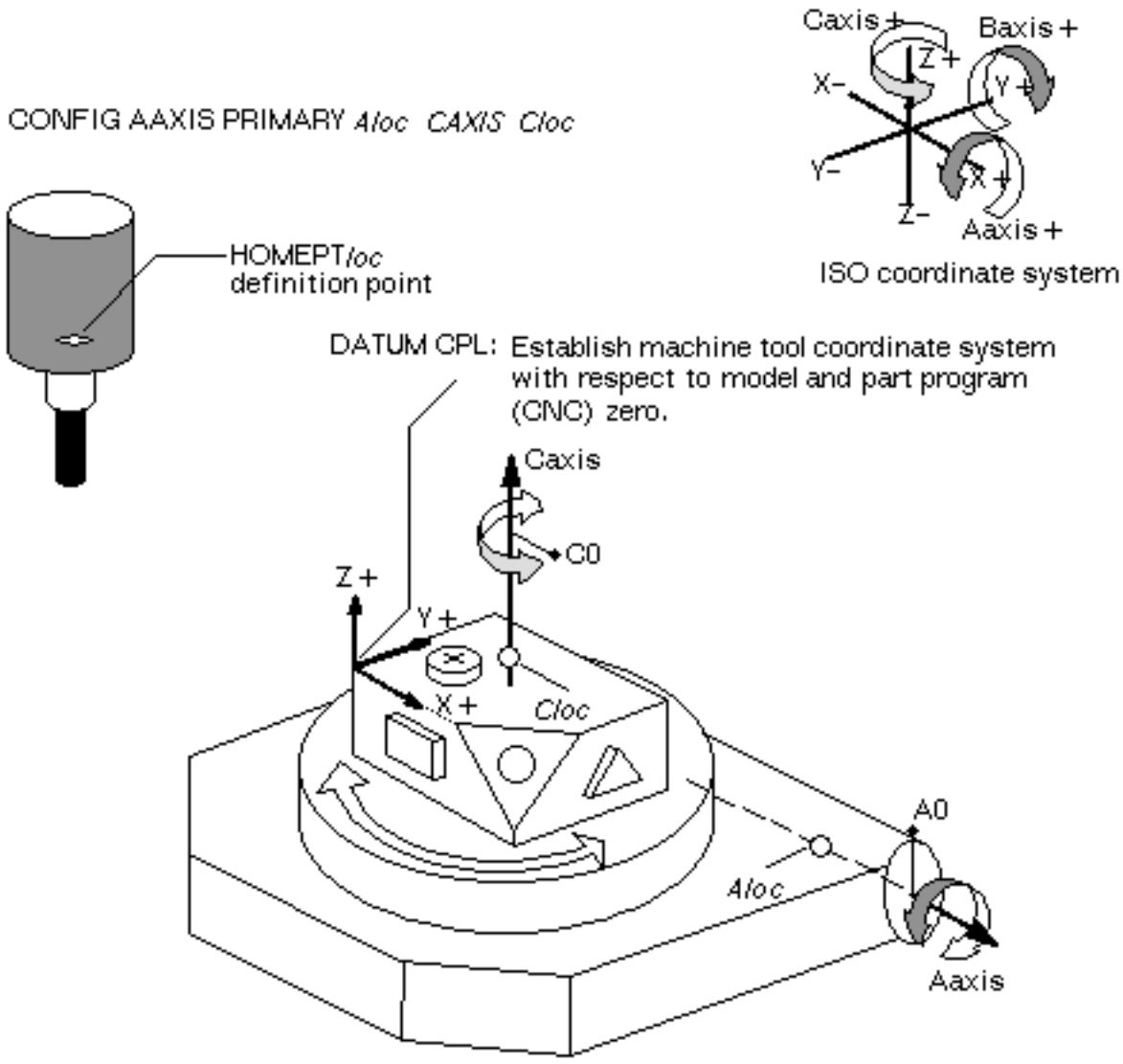
Use the following command format to define a 5-axis compound table machine:

```
NC:> CONFIG AAXIS PRIMARY MAXIMUM exp MINIMUM exp MININC exp loc
BAXIS MAXIMUM exp MINIMUM exp MININC exp loc
```

As stated previously, by omitting the word “HEAD” from the command, you indicate a table. Also notice that using the word “PRIMARY” indicates the relationship of the a-axis to the b-axis. You can think of it as “start with A and add B”.

The following figure shows you an example of the compound rotary table definition of a 5-axis milling machine. In this case, the c-axis is slaved to the a-axis, since rotation of the a-axis device affects the orientation of the c-axis table.

Figure 2-13 Compound Rotary Table Definition: 5-Axis Milling Machine

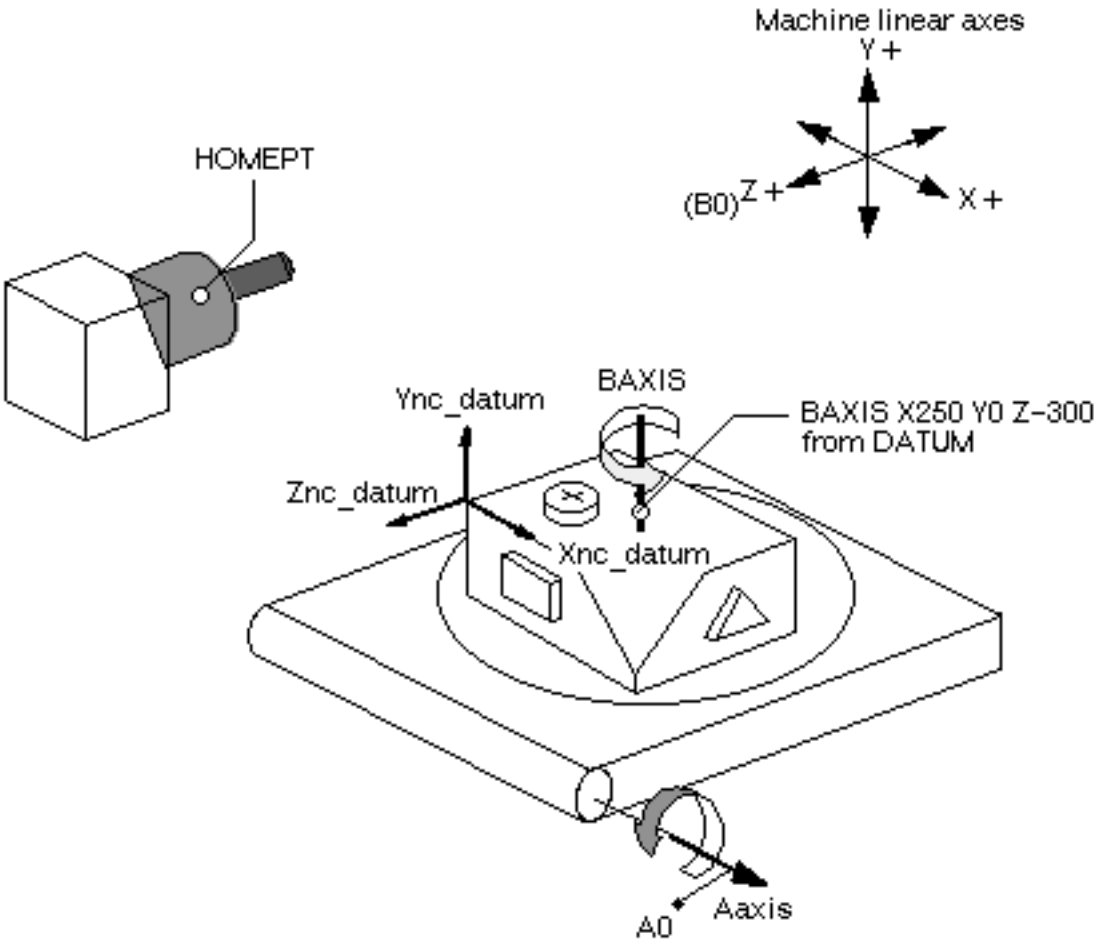



```
NC:> DATUM nc datum
NC:> HOMEPT FROM loc
NC:> CONFIG AAXIS PRIMARY Aloc CAXIS Cloc
NC:> CHGTOOL 1 'T1'
```

CAXIS is slaved to the AAXIS since if the AAXIS rotates, the orientation of the CAXIS table is affected.

The figure below shows how to configure a compound table where the a-axis is primary.

Figure 2-14 5-Axis Milling Machine: A-Axis and B-Axis Compound Table



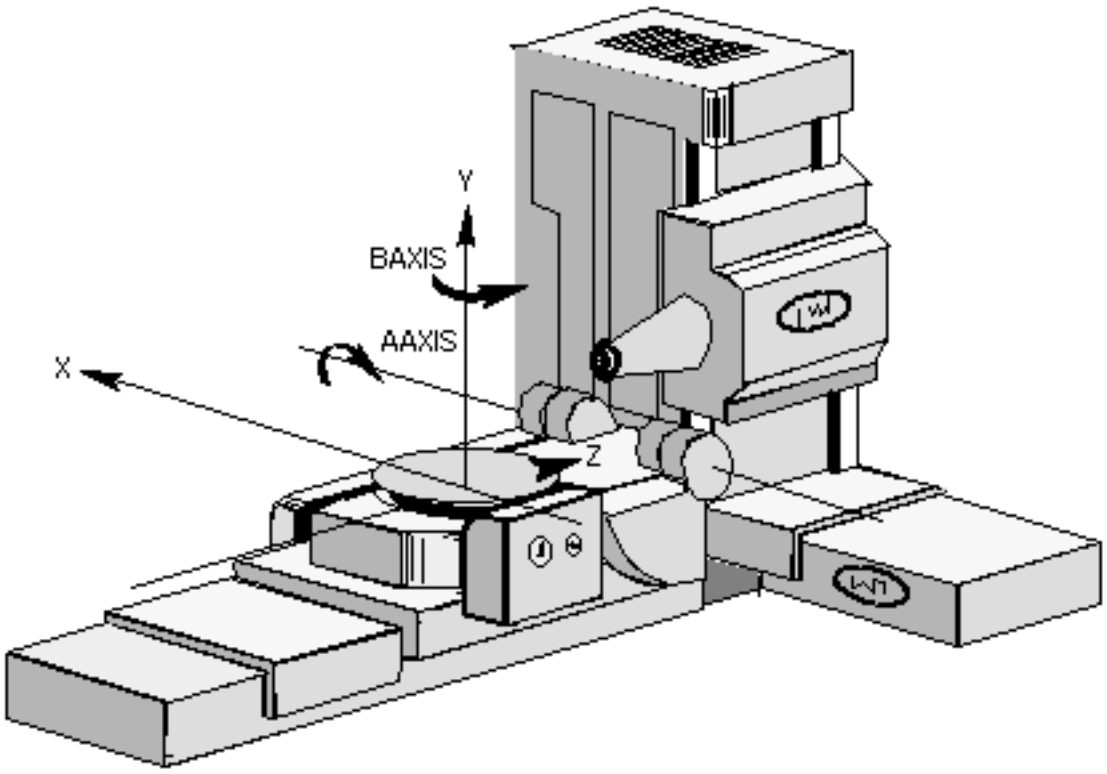

```
NC:> DATUM nc_datum
NC:> HOMEPT FROM X-100 Y200 Z500 ;
NC:> CONFIG  AAXIS PRIMARY Aloc BAXIS Bloc ;
NC:> CHGTOOL 1 '20MM S/DRILL'
```

The next figure furnishes an additional example of a milling machine with a primary a-axis rotary table and a secondary (slaved) b-axis rotary table.

You can configure this machine with the following command:

```
NC:> CONFIG AAXIS PRIMARY: Model loc d1 BAXIS: Model loc d2
```

Figure 2-15 Machine with A-Axis and B-Axis Rotary Tables



Configuring 5-Axis Milling Machines: Head and Table

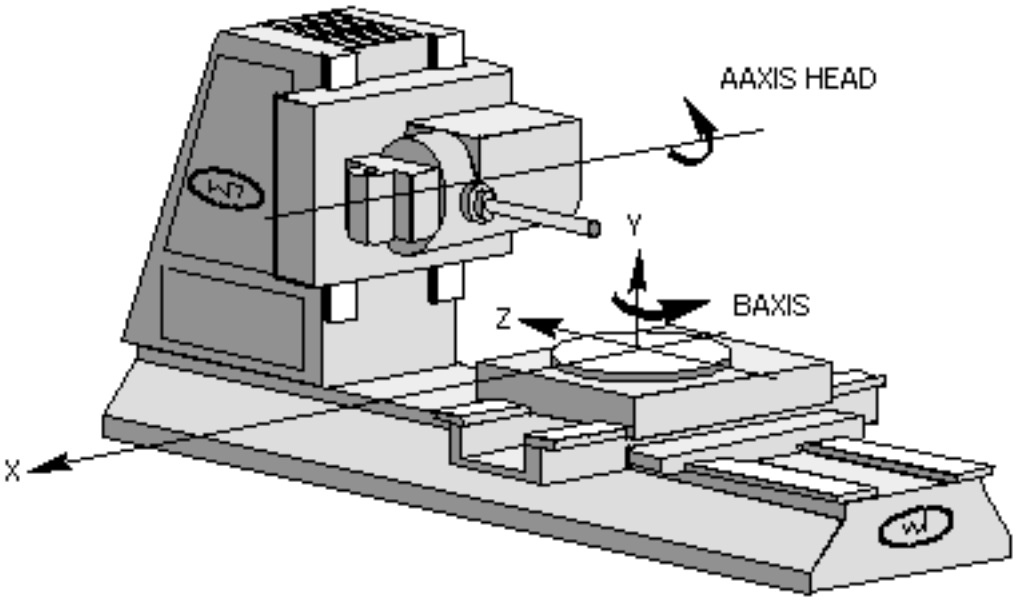
Use CONFIG to configure a 5-axis milling machine with a rotary head and a rotary table.

With the CONFIG command, you can configure a 5-axis milling machine with a rotary head and a rotary table.

The figure below shows a machine with a b-axis rotary table and an a-axis rotary head. You can configure it with this command:

```
NC:> CONFIG AAXIS HEAD: Model loc d1 BAXIS: Model loc d2
```

Figure 2-16 Machine with A-Axis Rotary Head and B-Axis Rotary Table



Using the HZERO Modifier in Configuring Rotary Heads

HZERO outputs coordinate data that relates to the tool location of the nonindexed position regardless of orientation.

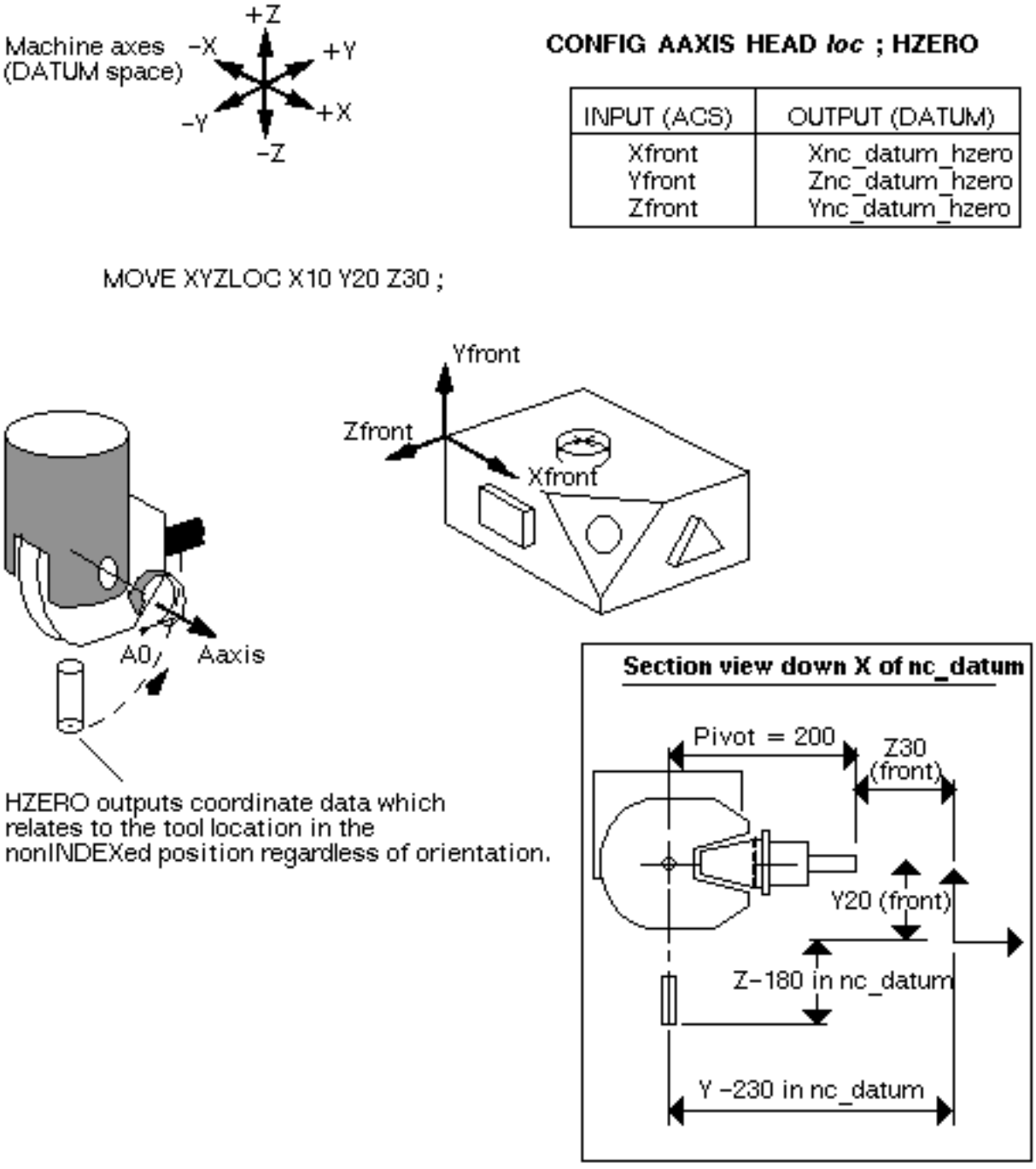
Use the CONFIG command's HZERO modifier to force the coordinates generated in the output file to be those that represent the tool when in the initial position, which is no rotation. This only applies to head machines.

The table below shows how to use the HZERO modifier in a sample sequence of commands.

Jcf Command	Action / Results	CLFILE OUTPUT
NC: > **DATUM 'TOP'**	Set machine coord system	None
NC: > **HOMEPT FROM X-100 Y-250 Z300 ;**	Establish gage line locn with respect to DATUM.	FROM / -100 -250 300
NC: > **CONFIG AAXIS** HEAD X-100 Y-250 Z400 ; **HZERO**	Define machine rotary axis CONF Guration : pivot distance is 100 along Z. Specify OUTPUT to be nonINDEXed locn.	None
NC: > **CHGTOOL 1 '20MM SLOT DRILL'**	Load tool : gage length = 100 Tooltip to pivot axes in Z = 200	LOADTL / 1 LENGTH 100.0 OSETNO 1
NC: > **SHOW #CURLOC; => X-100 Y-250 Z200**	HOMEFT - #TLGAGE : tool alligned with spindle axis	None
NC: > **INDEX 'FRONT'**	The AAXIS rotates 90 degrees ACS - FRONT is selected	ROTATE AAXIS 90 CCLW
NC: > **SHOW #CURLOC; = > X-100 Y400 Z50**	Coord in ACS = FRONT	None
NC: > **MOVE XYZLOC X10 Y20 Z30 ;**	XYZ in ACS - FRONT OUTPUT HZERO pcint in DATUM axes	RAPID / GOTO / 10.0, -230, -180

The following figure illustrates the above commands.

Figure 2-17 The HZERO Option Used with Rotary Head Machines



Characteristics of the CONFIG Command

The CONFIG command has certain characteristics you should take into consideration when programming.

The CONFIG command has the following characteristics:

- There are no limits on rotary devices. For example, AAXIS is only capable of +/- 105 degrees.
- The rotary axis must lie in a machine tool plane: XY or XZ or YZ.
- No graphics are generated for rotary motion.

Invalid Configurations

The following uses of CONFIG constitute invalid machine types:

```
CONFIG BAXIS HEAD CAXIS HEAD....

CONFIG AAXIS HEAD CAXIS HEAD....
```

Both the above configurations are invalid within CVNC since the CAXIS (if possible) would rotate around the spindle axis and hence be of no practical value.

Unsupported Configurations

Although 5-axis AAXIS and BAXIS compound head milling machines do exist, CVNC does not support these configurations:

```
CONFIG AAXIS HEAD BAXIS HEAD....

CONFIG BAXIS HEAD AAXIS HEAD....
```

Chapter 3

Job Setup: Tool Definition

After you use the machine configuration commands, DATUM, HOMEPT FROM, and CONFIG, define your tools for the job with TLIB, DEFTOOL, and LISTOOL. Use DELTOOL to delete tools not needed.

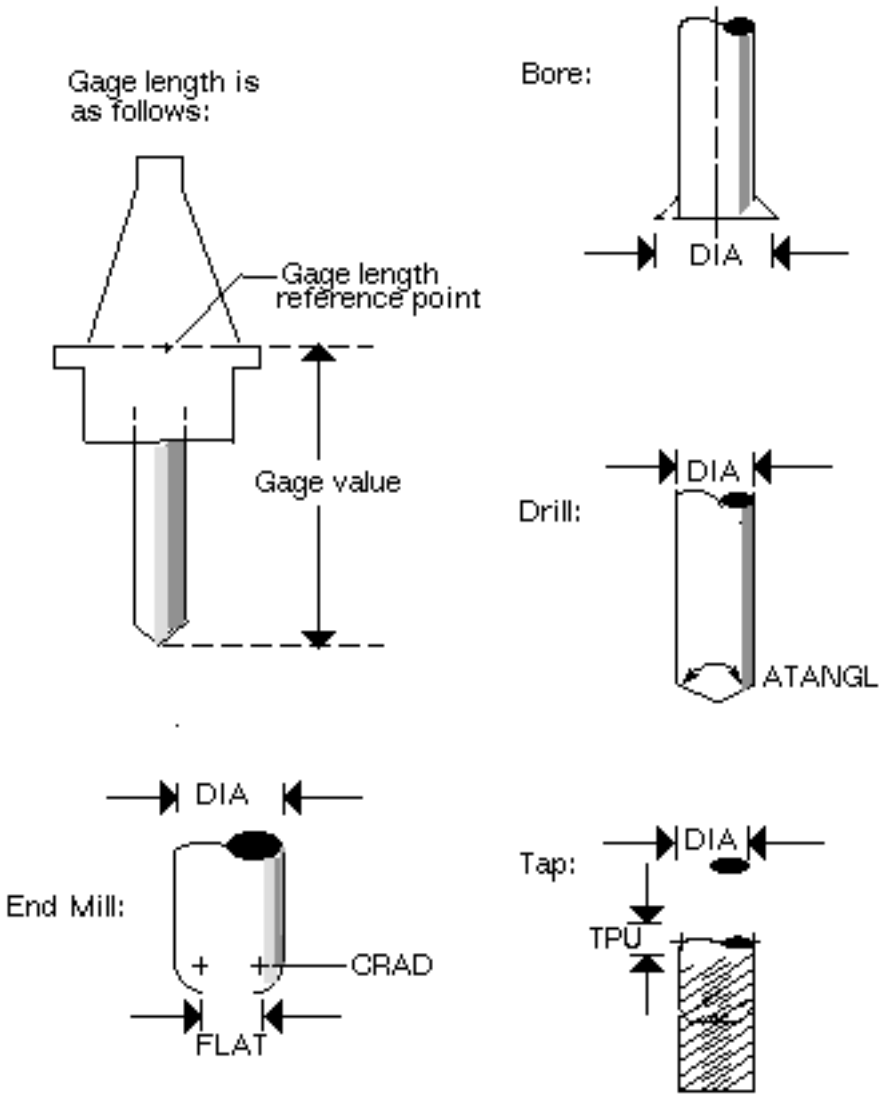
- Tool Definition Overview
- Setting Up Tool Libraries — TLIB
- Defining Tools — DEFTOOL
- Listing and Deleting Tools — LISTOOL and DELTOOL

Tool Definition Overview

Define your tools for the job with TLIB, DEFTOOL, LISTOOL, and DELTOOL.

CVNC-M2 supports milling, boring, drilling and tapping tools. The following figure shows typical cutting tools available for these functions. A milling tool example is shown later in this chapter.

Figure 3-1 CVNC-M2 Cutting Tool Examples: Bore, Drill, and Tap



Setting Up Tool Libraries — TLIB

Use TLIB to set up a tool library for a specific job or for general use.

CVNC provides a default tool library called `data/nc/tlib`. Once you define a tool in `data/nc/tlib`, you do not need to redefine it. No tools exist by default in this library.

To set up your own tool libraries, use the TLIB command. This gives you the flexibility to set up either a general library containing all the cutting tools used on your NC machine or several job-specific libraries.

If you use the general tool library to specify tools for a part program, you do not need to redefine the tool parameters for each JCF. If you create job-specific tool libraries, you must specify the tool library in each JCF.

For example, to set up a tool library called HEXTUL, enter

```
NC:> TLIB "HEXTUL"
```

This tool library then resides under `data/nc/hextul`.

To specify the tool library directory path and name, enter

```
NC:> TLIB =USERS.DATAPATH.TOOLSDIR."LIBNAME"
```

In this case, the equal sign (**=**) specifies the start of the top-level directory in a path. It is equivalent to the (**/**) root directory symbol in SunOS. The period (**.**) separates the directory name from the tool library name.

You can view the contents of your libraries using LISTOOL.

Defining Tools — DEFTOOL

The DEFTOOL command and its modifiers enable you to specify a range of tool geometries for your part.

Use DEFTOOL to specify a wide range of tool geometries for mills, bores, drills, and taps. The cutting tool parameters you define are stored in the current tool library.

Use the FLAT exp modifier with DEFTOOL to support CSINK (countersink) in a hole processing operation with a milling tool. Use the FLAT exp parameter to define the flat end of an angular cutting tool.

Please note: Use DEFTOOL to define a tool before you use CHGTOOL to specify that the tool be used to calculate tool motion.

Use the CORNER modifier to set the radius of a radial-end mill.

CVNC-M2 supports four types of mill ends, shown below. Examples of milling tools with detailed views of the tool ends are shown in the figures.

Figure 3-2 CVNC Mill Ends

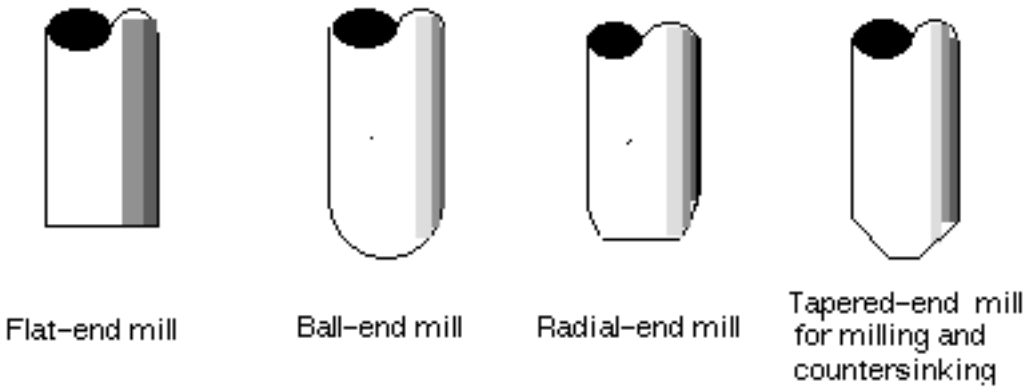


Figure 3-3 Milling Tool with a Corner Radius

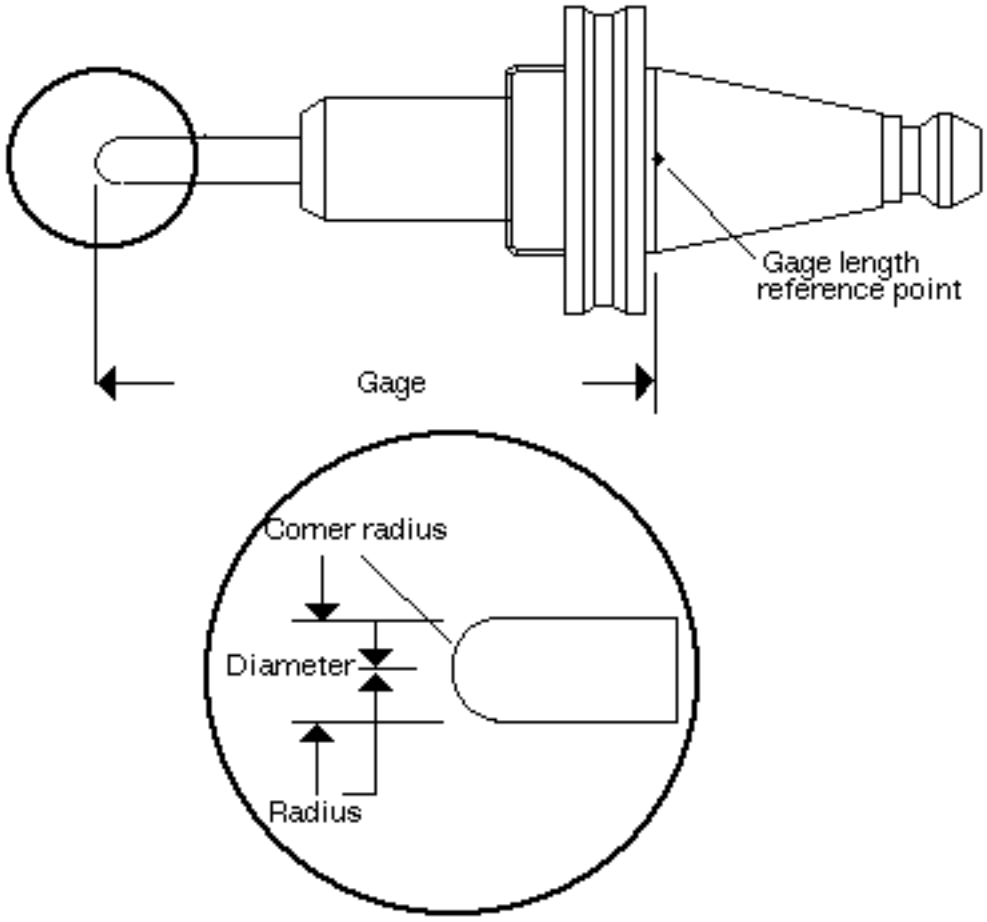


Figure 3-4 Milling Tool with a Flat End

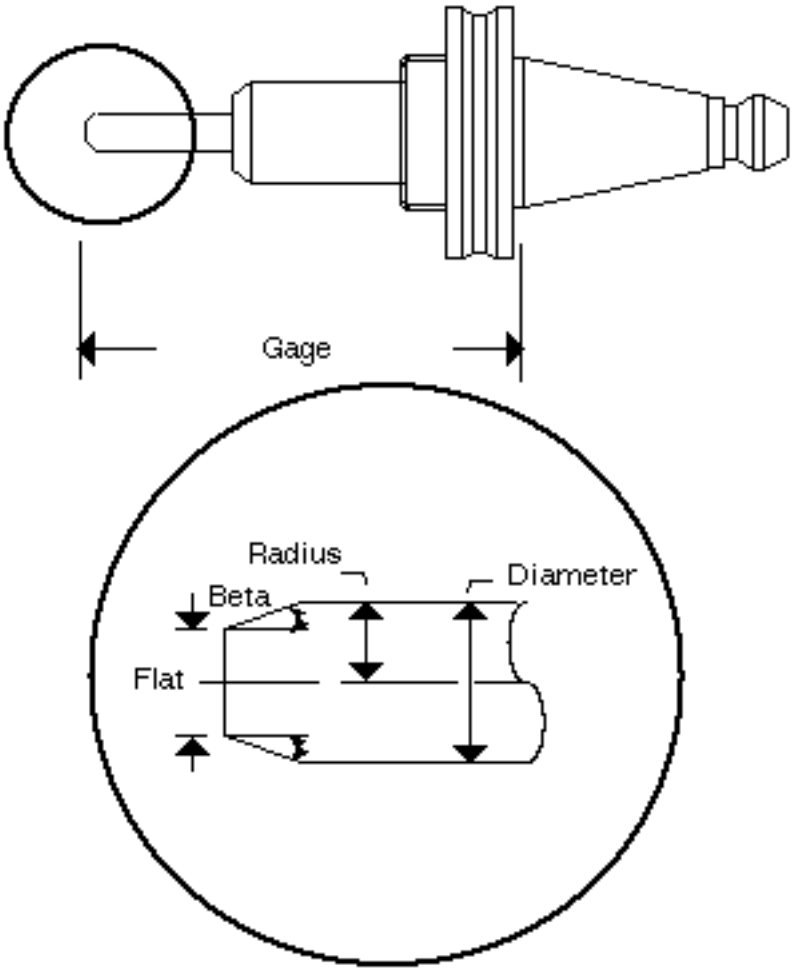
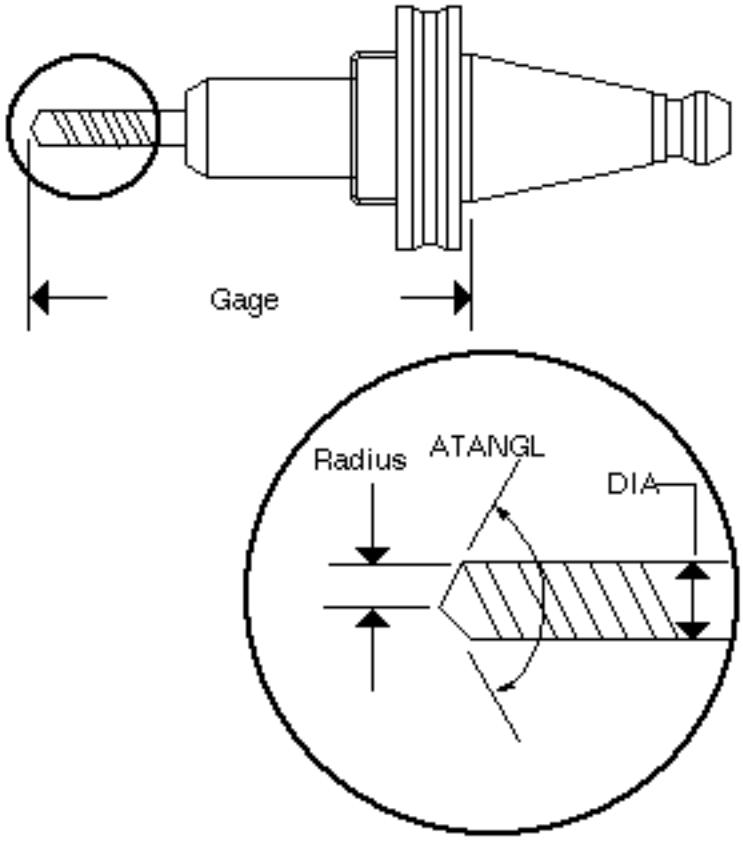


Figure 3-5 Standard Drill



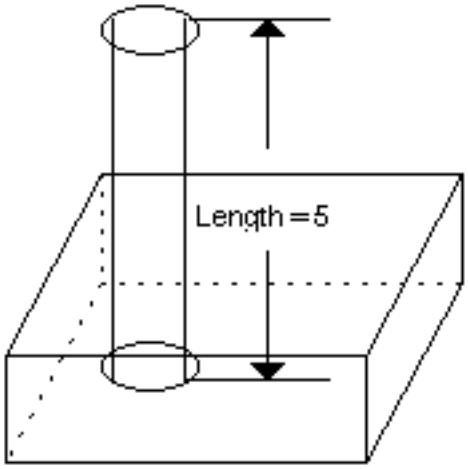
Please note: With a drill, use the ATANGL modifier to define the included drill point angle as shown in the next figure. With a tap, use the TPU modifier to define the number of threads per unit of measure.

You can retrieve tool parameters by referencing the tool name with CHGTOOL.

Use the following sample sequence as a model. In this example, DEFTOOL defines a milling tool named “RAPPR”, as shown in the figure.

Figure 3-6 DISPLAY TOOL Default Gage Lengths

```
NC:>DEFTOOL 'RAPPR' MILL7 DIA 2.0 CORNER 0
HEIGHT 3 GAGE 5
NC:>CHGTOOL 1 "RAPPR"
NC:>DISPLAY TOOL DYN VIEW: vie d
NC:>PLUNGE
```



Defining a Countersink Tool

The countersinking operation (CSINK) supports the use of drills and both tapered and milling tools.

Define a tool for countersinking as a MILL (where the BETA value angle is half the countersink angle), or a DRILL (where the ATANGL value is the countersink angle).

Warning

Do not use the MILL7 modifier to define a countersink tool.

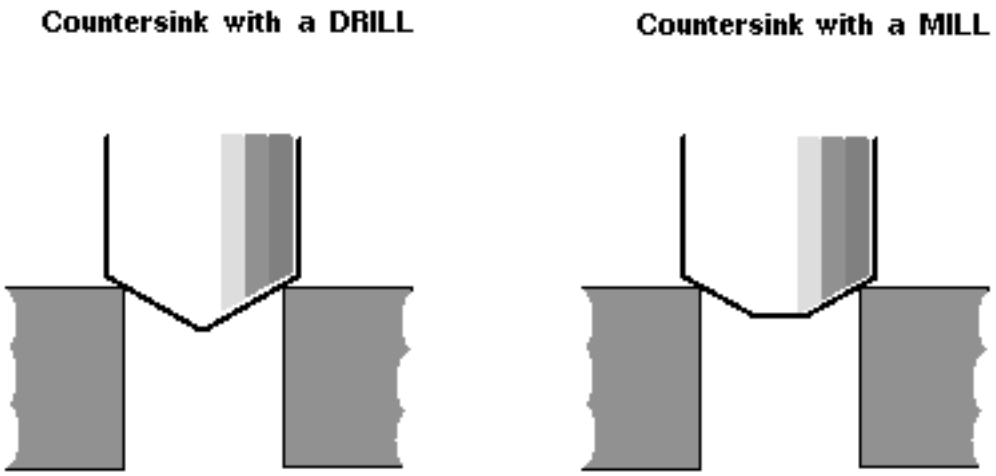
To machine a 120° inclusive chamfer using a DRILL, define the tool by entering

```
NC:> DEFTOOL '120 DEGREE CHAMFER TOOL' DRILL DIA 1 ATANGL 120
```

If you are using a tapered or milling tool, enter

```
NC:> DEFTOOL '120 DEGREE DRILL' MILL DIA 1 BETA 30 FLAT .25
```

Please note: Use a MILL definition rather than a DRILL when you want CVNC to calculate for a truncated countersink tool.



Defining 2-Parameter Tools with the MILL7 Modifier

Use the MILL7 modifier (rather than the MILL modifier) to define milling tools for CVNC-M2. If you use MILL, CVNC issues a warning, instructing you to use MILL7 instead.

Warning

When defining a countersinking tool, do not use the MILL7 modifier.

The following figure shows a milling tool example with a description of the tool's parts.

To define a 2-parameter tool with the MILL7 modifier, follow these steps:

1. Enter DEFTOOL and a tool description enclosed in either single or double quotation marks with the MILL7 modifier.
2. Specify the constant diameter (DIA) and the corner radius (CORNER).
3. Enter the cutting length of the tool (HEIGHT). Do not use the ECORNER, FCORNER, ALPHA, or BETA modifiers.
4. Specify the gage length (GAGE) and give a description (DESC followed by "text description"), if required.

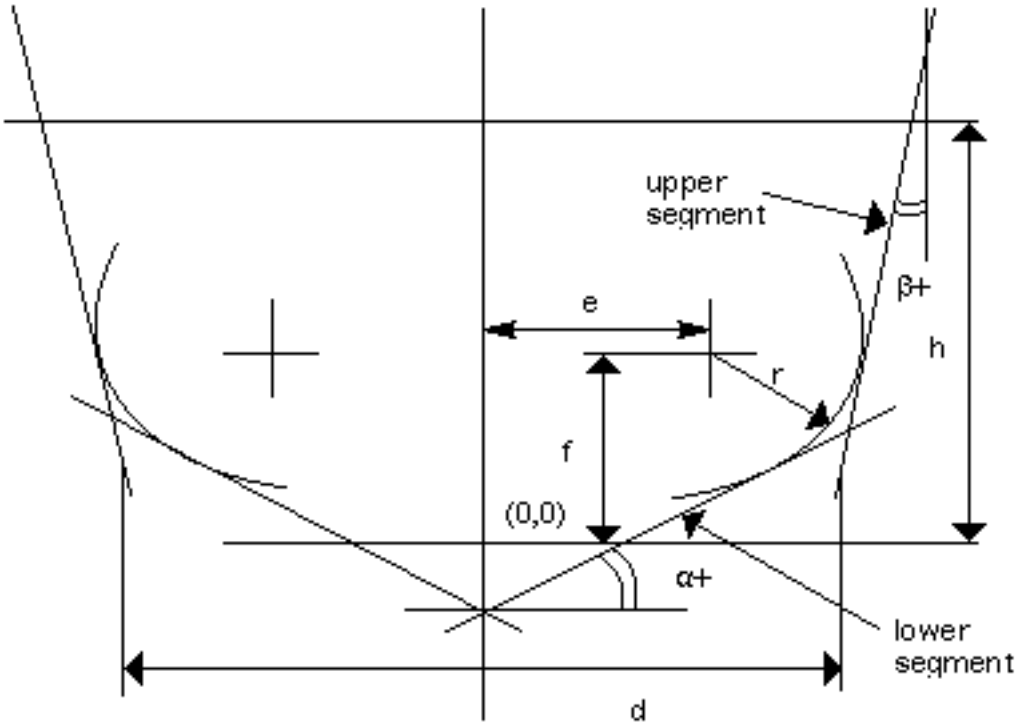
   For example:

```
NC:> DEFTOOL "KQ2" MILL7 DIA .5 CORNER .85 HEIGHT 2.75 GAGE 2.5
```

Please note: GAGE and HEIGHT define different distances. HEIGHT (a mandatory modifier) defines the cutting length of the tool. GAGE (an optional modifier) defines the distance between the tool tip and the gage reference point. If you do not specify a gage length, GAGE defaults to the length defined by the HEIGHT modifier.

Please note: Full 7-parameter tools must be used with CVNC-M3 only.

Figure 3-7 7-Parameter Milling Tool Example



The following table describes the relationship between the tool parameters and DEFTOOL modifiers.

Table 3-1

Tool Parameter	Description	DEFTOOL Modifier
d	Tool diameter at the intersection of the upper and lower segments	DIA
r	Tool corner radius between the upper and lower segments	CORNER
a	Angle of the lower segment with the tool radial axis	ALPHA
b	Angle of the upper segment with the tool vertical axis	BETA
f	Distance from the center of the corner radius to the bottom of the tool	FCORNER
e	Distance from the center of the corner radius to the tool axis	ECORNER
h	Distance from the bottom of the tool to the top of the tool (axial cutting length of the tool)	HEIGHT

Listing and Deleting Tools — LISTOOL and DELTOOL

This section describes the LISTOOL and DELTOOL commands.

LISTOOL

LISTOOL lists the parameters of one tool (or all tools) in the current or specified tool library. If the tool name is not specified, the command lists all tools in the current or specified library. LISTOOL does not become a permanent record in the JCF.

For example, to list the parameters of the tool called RAM in the tool library named HEXTUL, enter

```
NC:> LISTOOL "RAM" TLIB "HEXTUL"
```

DELTOOL

Use DELTOOL to delete a tool from your current tool library. For example, to delete the tool called RAM from your current library, enter

```
NC:> DELTOOL "RAM"
```

Please note: After you define a tool, you cannot modify its parameters unless you delete the tool (DELTOOL) and redefine it (DEFTOOL).

Chapter 4

Operation Setup: Regulating Cutting Operations

With the commands described in this chapter, you can set up and regulate cutting operations. With CHGTOOL, select the desired tool. Specify a new Cplane with CPL. Use INDEX to drive rotary devices.

- Overview of Regulating Cutting Operations
- Changing Tools — CHGTOOL
- Specifying a New Construction Plane — CPL
- Programming Rotary Axes — INDEX
- Using the CPL Command with Respect to INDEX
- Input to Output Coordinate System Relationships
- CVNC to NC Machine Relationships

Overview of Regulating Cutting Operations

With the commands described in this chapter, you can set up and regulate cutting operations.

Use CHGTOOL to select a tool for a particular operation. Use CPL to specify a new Cplane.

Use INDEX to move rotary devices so that the part and Cplane align with the machine coordinate system. After the rotation, use INDEX to activate the indexed Cplane in CADDS.

If you used CONFIG to specify the rotary devices, you can change from your current Cplane to another predefined Cplane by entering INDEX.

Changing Tools — CHGTOOL

Use CHGTOOL to select a cutting tool (predefined with DEFTOOL) from the active tool library for an individual milling or hole processing operation.

CHGTOOL modifiers enable you to specify

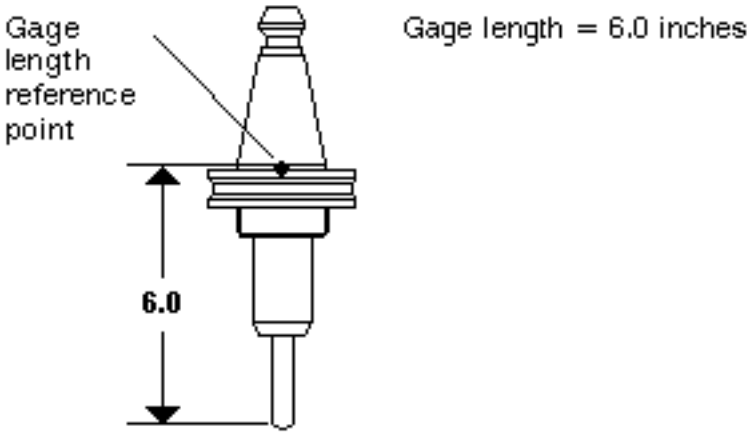
- Tool (station) number and library name
- Tool diameter and tool length compensation registers
- Manual or automatic change
- Directional indexing of the turret or carousel
- Part name of the associated SFIGURE (Sfigure file)

You can also prevent the generation of output for this command.

To change to the tool “1 INCH DRILL”, with a gage length of 6 inches, enter

```
NC:> DEFTOOL "1 INCH DRILL" DRILL DIA 1 GAGE 6.0
NC:> CHGTOOL 1 "1 INCH DRILL"
```

Figure 4-1 Sample Tool “1 INCH DRILL”



Specifying a New Construction Plane — CPL

CPL lets you select the optimum input coordinate system (xyz) for the machining sequence being programmed.

When you specify a Cplane by name as the coordinate system, all JCF xyz coordinates will be interpreted with respect to the named Cplane.

The following example illustrates use of CPL. The figure shows a typical aerospace component that, due to grain flow requirements (and/or size), must be programmed at an angle with respect to the rectangular piece of material. The material is loaded onto the machine bed parallel to the machine axes.

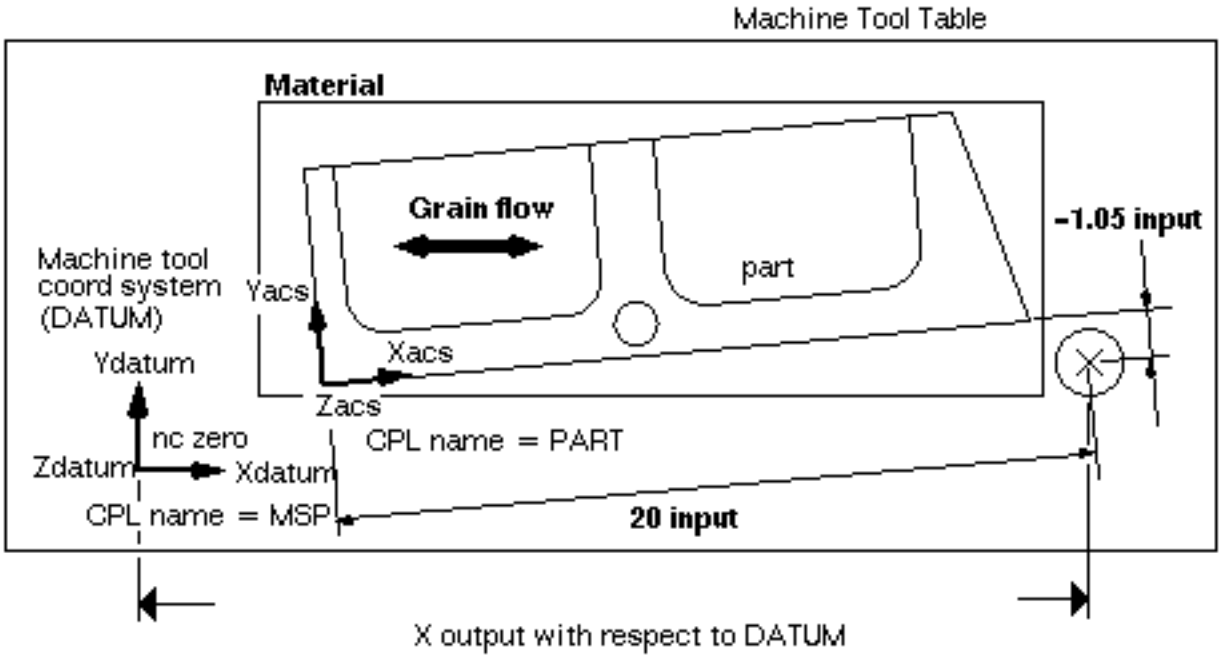
Commands in the lower-left corner show you how to program this sequence.

In the setup shown in the following figure, the material is aligned with the machine tool and the component has a specified grain flow direction with respect to the part.

DATUM is used to specify the machine coordinate system local to the part. This allows you to change the orientation of the part with respect to the machine tool/material without having to edit all the xyz coordinates in the JCF. Such an occasion might be due to a need for different fixtures, for example.

All xyz data is mapped back to the new DATUM in the output file.

Figure 4-2 Using the CPL Command



In the previous figure, DATUM specifies the machine tool coordinate system. Use CPL to select a work coordinate system convenient for machining the part geometry. The xy-coordinate in the MOVE command is thus measured from CPL "PART" and output is mapped back to DATUM.

Selecting or Creating a New Cplane

Specify a new Cplane or create one from your active Cplane with CPL by giving the new Cplane location relative to the present Cplane. With CPL modifiers, you can

- Select a new Cplane. For example,

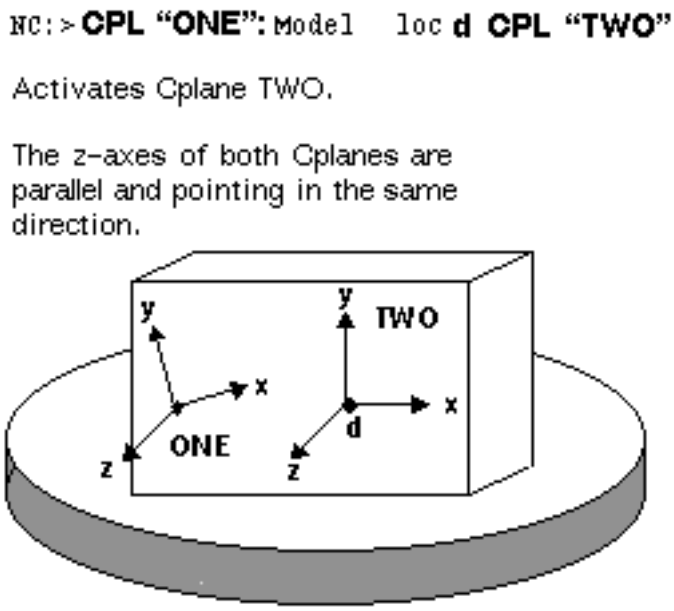
```
NC:> CPL "TOP"
```

- Specify units along the x-, y-, or z-axis to offset the existing Cplane. For example, to offset a Cplane named FRONT by one unit along the x-axis and one unit along the y-axis, enter

```
NC:> CPL "FRONT" XCPL 1 YCPL 1 CPL "FRONT-X1Y1"
```

- The new Cplane is named FRONT-X1Y1.
- Specify the origin of the new Cplane, as shown in the example given below.

Figure 4-3 Digitizing a New Cplane



If you specify a new Cplane name without the offset modifiers (XCPL, YCPL, and ZCPL) or a location (loc), the result is two identical Cplanes with different names. For example,

```
NC:> CPL "TOP" CPL "NEWTOP"
```

Restricting Cplane Selection to a Constant Z

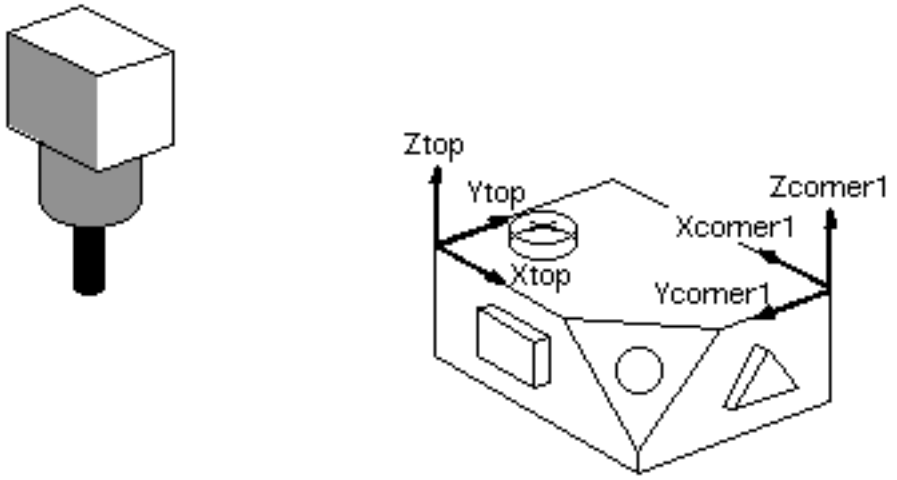
When using CPL, you can only select a new Cplane in which the z-axis is the same as the active Cplane.

Since CPL does not drive rotary axes or change the tool axis with respect to the part, you are restricted in the selection of a new Cplane to one in which the z-axis is same as the active Cplane.

The following figure shows examples of valid and invalid CPL commands. If you select TOP as DATUM, CPL "FRONT" is invalid because to align the tool axis with the z-axis of CPL FRONT requires a rotary axis. Any Cplane coplanar in TOP would be valid.

For a 3-axis machine tool, you can select Cplanes in the DATUM plane only, as shown in the following figure. An attempt to reorient the tool axis using CPL is trapped.

Figure 4-4 Validity of Cplanes




```
NC:> DATUM 'TOP'
NC:> HOMEPT FROM X-100 Y200 Z500 ;
NC:> CHGTOOL 1 '20MM S/DRILL'
NC:> CPL 'CORNER1' ———————— valid
NC:> CPL 'FRONT' ———————— invalid
```

Programming Rotary Axes — INDEX

Use INDEX to program rotary heads and/or tables to align your tool axis normal to the work plane.

Use INDEX to drive rotary devices.

You can program (preconfigured with CONFIG) rotary heads and/or tables in 2 1/2-axis mode (rotation only, not 4- or 5-axis motion) by using INDEX. These are commonly termed prismatic applications.

The applications align the tool axis normal to the 2D plane in which subsequent machining is to be performed.

The following rules apply to rotary motion:

- The right hand screw rule applies to all rotary motion, that is, a negative angle generates a clockwise motion and vice versa.
- In CVNC, it is always the tool that moves and not the table. This is irrespective of the type of device. It is important to take this into consideration while working in CVNC.
- For a head device a positive angle in CVNC corresponds to a positive angle in the NC machine head.
- For a table device a positive angle in CVNC corresponds to a negative angle in the NC machine table.

Before using INDEX, configure the target machine tool. The machine tool definition then provides information needed for such functions as validity checking, where it ascertains that the orientation is possible with the axes defined.

There are four ways of programming rotary axes within CVNC:

- Specifying the axes
- Using a Cplane
- Using a normal
- Using a plane

If you specify an axis (AAXIS, BAXIS, or CAXIS) with INDEX, you can specify the angle of rotation in either absolute or incremental terms.

For example, you can use INDEX in any of these ways:

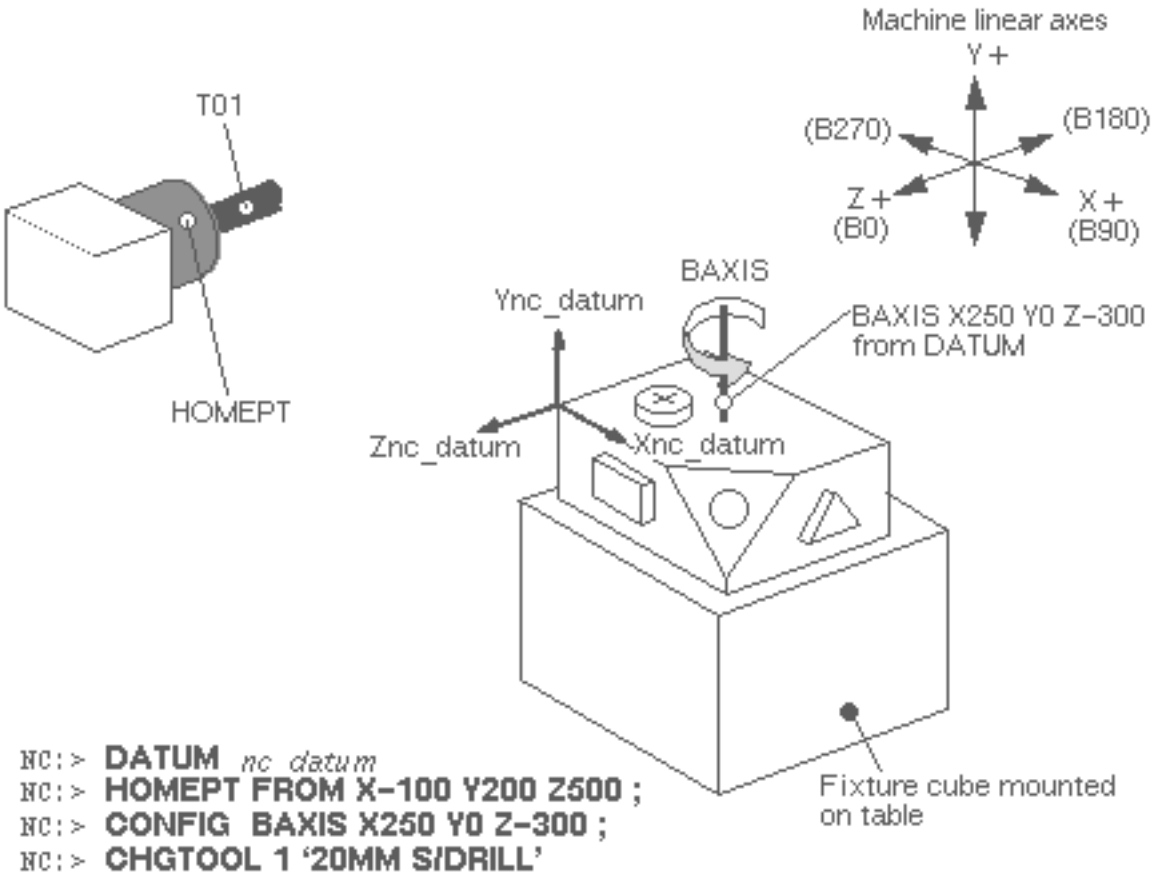
```
NC:> INDEX BAXIS 90 (explicit axis, incremented)
NC:> INDEX "RIGHT" (RIGHT is the name of a predefined CADDS
Cplane)
NC:> INDEX NORMAL
NC:> INDEX PLANE
```

The configured machine tool provides CVNC with the information to handle functions such as tool-to-part relationship and input-to-output coordinate mapping.

The following figure shows a typical component mounted on a 4-axis, horizontal b-axis table, milling machine in which machining of multiple orientations is to be performed in a single setup.

This example explains the function of INDEX. The diagram shows the true machine tool view.

Figure 4-5 4-Axis, Horizontal B-Axis Table Milling Machine



DATUM defines the machine tool coordinate system with respect to the model.

CONFIG defines a b-axis table with rotational center at X250 Y0 Z-300 with respect to the DATUM CPL. When a tool is loaded, its axis is aligned with the z-axis of DATUM. Then the required machining sequences are performed; for example, profiling the rectangle.

To profile the triangle, the b-axis must be rotated through 90°, rotating all elements mounted on the table. The center of rotation is around the true position of the b-axis at the time of rotation.

You do not need to specify whether the head or table (or both) is being rotated. The machining requirement is to position the tool in relation to the part.

Knowing the machine configuration, CVNC rotates the head and/or table to the specified destination. CVNC also resolves the position of the tool with respect to the part after using INDEX.

Input/Output coordinate mapping depends on whether head or table rotation is being made and is resolved by CVNC.

Specifying Axes Directly

Use INDEX to specify rotation around the a-, b-, or c-axis.

After you have defined the rotary axes of the target machine tool, you can define orientation by the AAXIS address (A, B, or C) followed by its position in decimal degrees.

Using this form of the command, you can rotate only one axis at a time. If the desired orientation requires that two rotary axes be rotated, you must use two INDEX commands.

To profile the triangle in the previous figure, enter

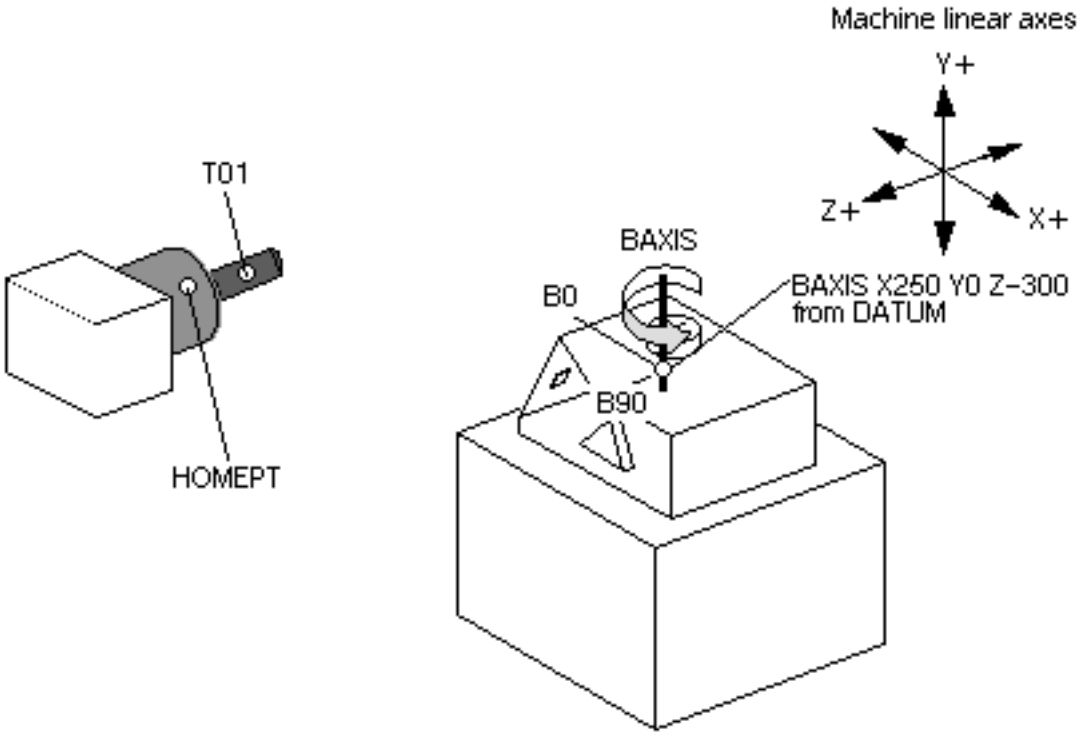
```
NC:> INDEX BAXIS 90 CCLW
```

In this example, note that the option to specify counterclockwise (CCLW) rotation has been chosen. This was determined by considering the cutting tool analogous to linear motion that specifies x+ and y+. However, in the case of a table machine, the table moves in the opposite direction.

The result of INDEX executed on the figure on the next page is shown in the figure, CVNC Interpretation of Rotating a Table, on the subsequent page.

Please note: You may see additional statements in your output file, such as ROTATE/ AAXIS, ATANGL, 0.0000. CCLW, when using the INDEX command. Ignore these statements as they do not affect output.

Figure 4-6 4-Axis Horizontal B-Axis Table Milling Machine

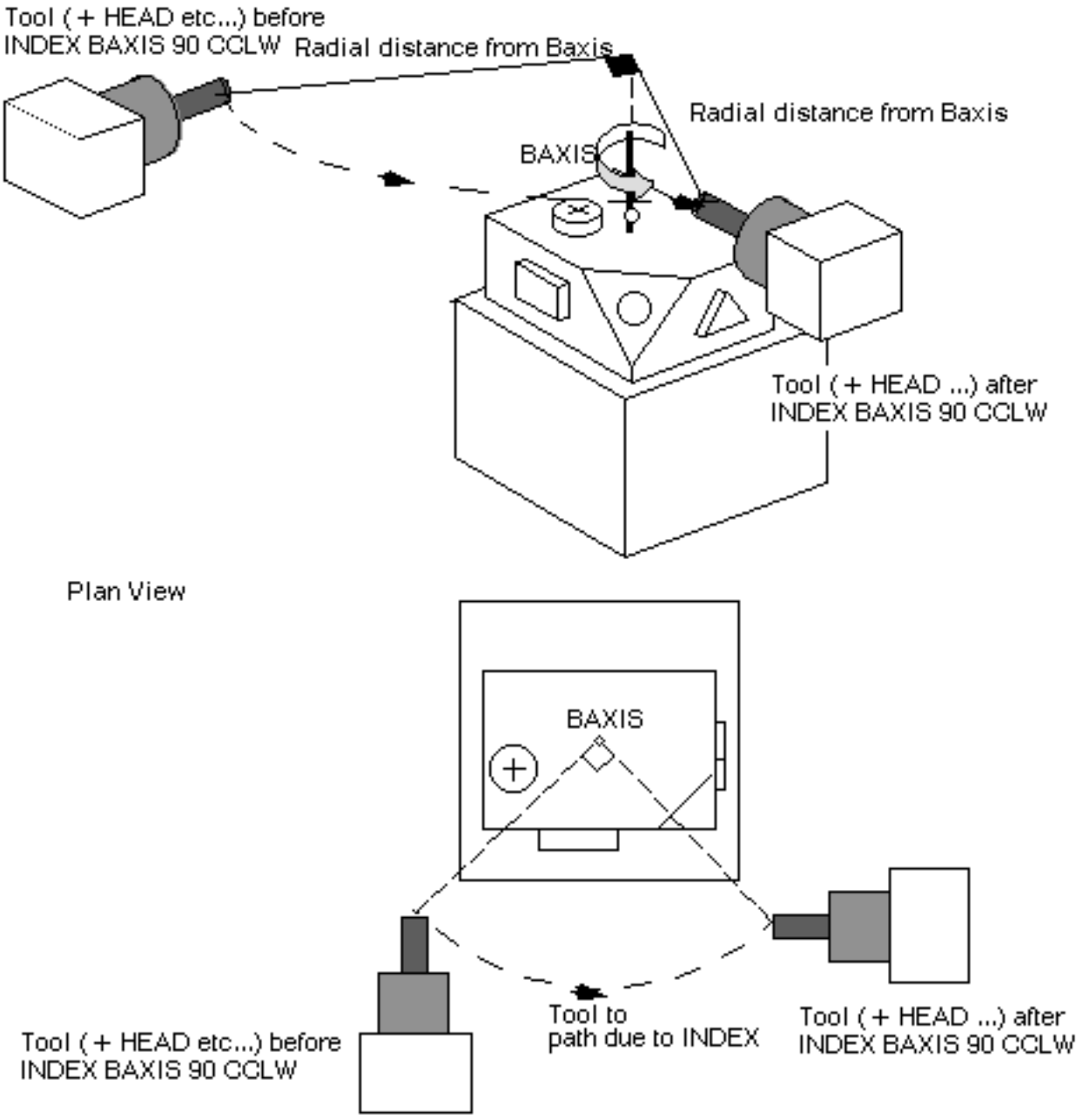



```
NC:> DATUM nc datum
NC:> HOMEPT FROM X-100 Y200 Z500 ;
NC:> CONFIG  BAXIS X250 Y0 Z-300 ;
NC:> CHGTOOL 1 '20MM S/DRILL'
NC:> INDEX BAXIS 90 CCLW
```

In the previous figure, the triangle moves into the xy-plane of the machine tool.

The graphical simulation produced in CVNC illustrated in the previous figure does not show rotation of the part (fixture, cube, etc.). Instead, it shows all elements of the machine tool (head, tool, etc.) rotated around the part. This is illustrated in the following figure.

Figure 4-7 CVNC Interpretation of Rotating a Table



Rotating the machine around the part occurs for rotary table configurations only. CVNC does not show the connection arc path graphics. Thus INDEX causes the tool image to disappear at its current tool location and reappear in the new position.

Using a Cplane to Define Orientation

Use a Cplane to define the new orientation for a machining operation.

To use a Cplane to define the new orientation necessary for subsequent machining, use INDEX in this format:

```
NC:> INDEX "Cplanename"
```

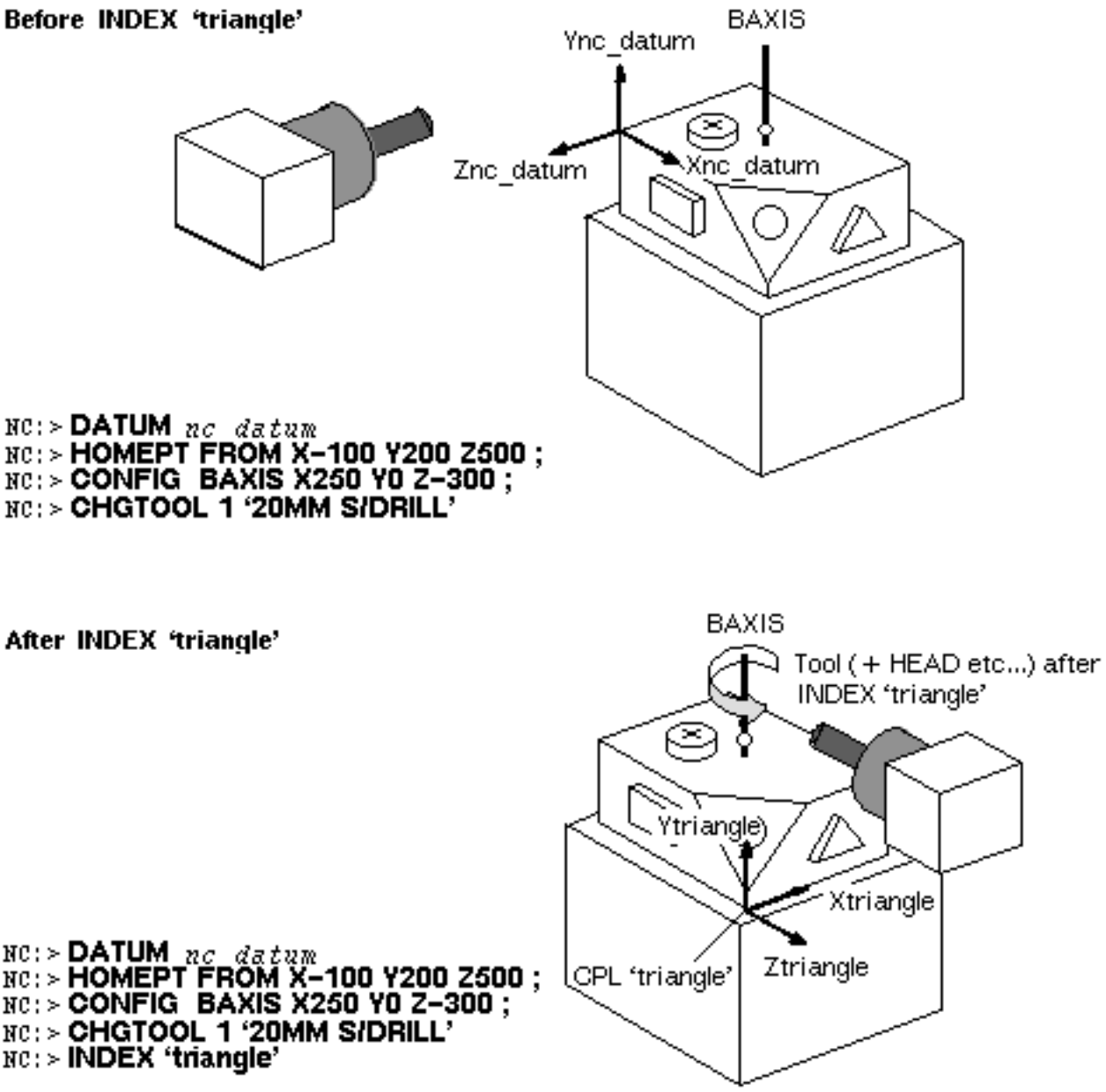
where,

`Cplanename` is the Cplane that defines the new orientation.

CVNC aligns the tool axis with the z-axis of the specified Cplane, using the rotary axes defined in CONFIG.

The following figure shows how, after defining a Cplane relative to the triangle on the part, you can then use that Cplane to define the required orientation.

Figure 4-8 Use of Cplanename to Drive Rotary Axes (CVNC graphics)



The Cplane named TRIANGLE has an origin and orientation inherent in its definition in 3D space. The ACS after INDEX becomes Cplane TRIANGLE.

You cannot specify a direction of rotation when using `Cplanename` to drive rotary axes. CVNC assumes CCLW, which is written to any output file generated with the ROTATE record.

Using a Normal to Align the Tool Axis

Use INDEX to align the machine tool axis with a specified normal in this format:

```
NC:> INDEX NORMAL loc loc
```

The first `loc` indicates the base of the normal and the second indicates the direction of the positive z-axis.

You have to digitize at both `locs`. CVNC creates a new Cplane with the first `loc` as its origin. You can specify both the name of the new Cplane and the x-, y-, and z-offsets for its origin.

Please note: The two `locs` should define a unique line.

Using a Plane

You can align the machine tool axis normal to a specified plane. CVNC defines and creates a Cplane at the indicated orientation. You have to digitize at three locations. Use INDEX in the following format:

```
NC:> INDEX PLANE loc loc loc
```

where the three `locs` indicate the origin of the new Cplane, the direction of the positive x-axis and the relative direction of the positive y-axis, respectively.

You can specify the name of the new Cplane and the x-, y- and z-offsets for its origin.

Please note: The three `locs` should define a unique plane.

System-generated Cplanes

A new Cplane is generated when you specify rotation with the INDEX command, using the explicit axis format.

CVNC requires an input coordinate system defined by a Cplane to be active at any time.

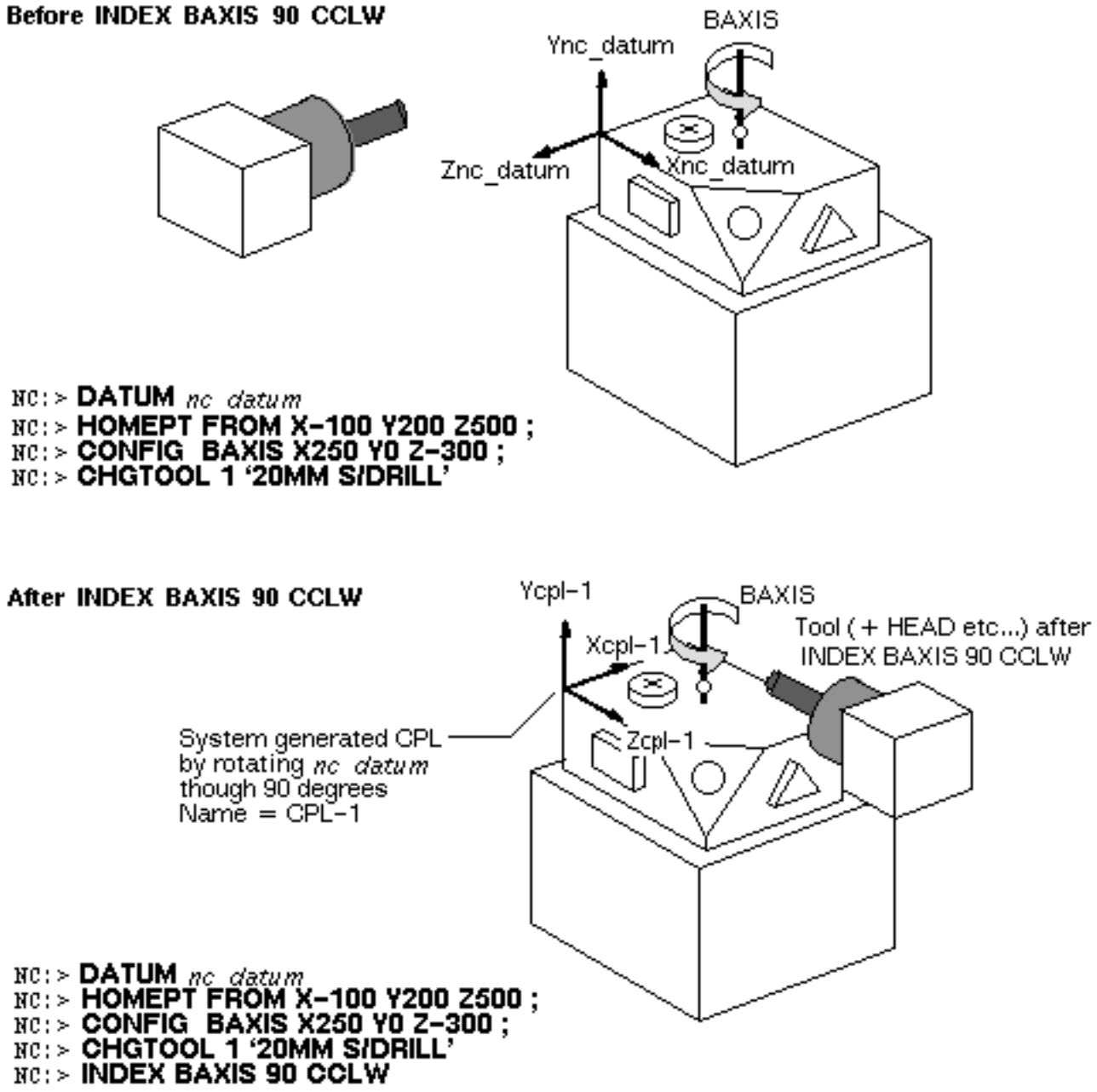
When you specify rotation by using the explicit axis format (AAXIS, BAXIS, CAXIS) with INDEX, CVNC generates a Cplane. This Cplane then becomes the ACS (Active Construction Space). Hence all subsequent motion is with respect to this Cplane.

As a naming convention for these system-generated Cplanes, CVNC uses CPL-nexp. Thus for a new part, a Cplane named Cplane-1 is generated when the INDEX BAXIS 90 CCLW command is issued, as shown in the example in the earlier section, “Specifying Axes Directly” on page 4-10.

The origin of a system-generated Cplane is the same as that of the Cplane current at the time you issued the INDEX command. The orientation is determined by rotating the ACS in accordance with the axis and amount specified in the INDEX command.

In the following figure, Cplane-1 is generated by rotating the Cplane named “NC_DATUM” (set up by DATUM) through 90° around the b-axis.

Figure 4-9 System-generated Cplane for INDEX BAXIS 90 CCLW (CVNC Graphics)



In the previous example, INDEX generates Cplane-1, which shares its origin with the Cplane called “NC_DATUM”.

The tool axis always aligns with the z-axis of the ACS. You can then program in a convenient input coordinate system.

The relationship to machine tool axes is resolved by CVNC.

Validating INDEX

CVNC validates the INDEX command with three levels of testing.

When you enter INDEX, CVNC validates the input against the machine tool rotary axis capability and issues appropriate information messages or error conditions.

When a Cplane defines the target orientation, three levels of testing and message generation occur:

1. Can the DATUM CPL be fully rotated to align with the target Cplane in x, y, and z?

   If this is possible, the command is accepted and you receive this message:

```
Cplane Activated: Tool coordinate system rotated to align
with CPL.
```

   If this is not possible, the test is made.

2. Can the tool axis be aligned with the z-axis only of the target Cplane?

   If this is possible, the command is accepted and you receive this message:

```
Cplane Activated: Tool axis only aligned with ZAXIS of
requested CPL.
```

   If this is not possible, the orientation cannot be achieved.

3. Hence an error condition is produced because the configured machine tool cannot be programmed to the requested orientation. You receive this message:

```
Requested CPL cannot be attained with the current
configuration.
```

Fully Aligning Rotary Axes with the Cplane — NOOPTIM Modifier

CVNC attempts to align the DATUM xyz coordinates with the target Cplane xyz coordinates using the rotary axes defined when you use the INDEX NOOPTIM modifier. This alignment may not be possible.

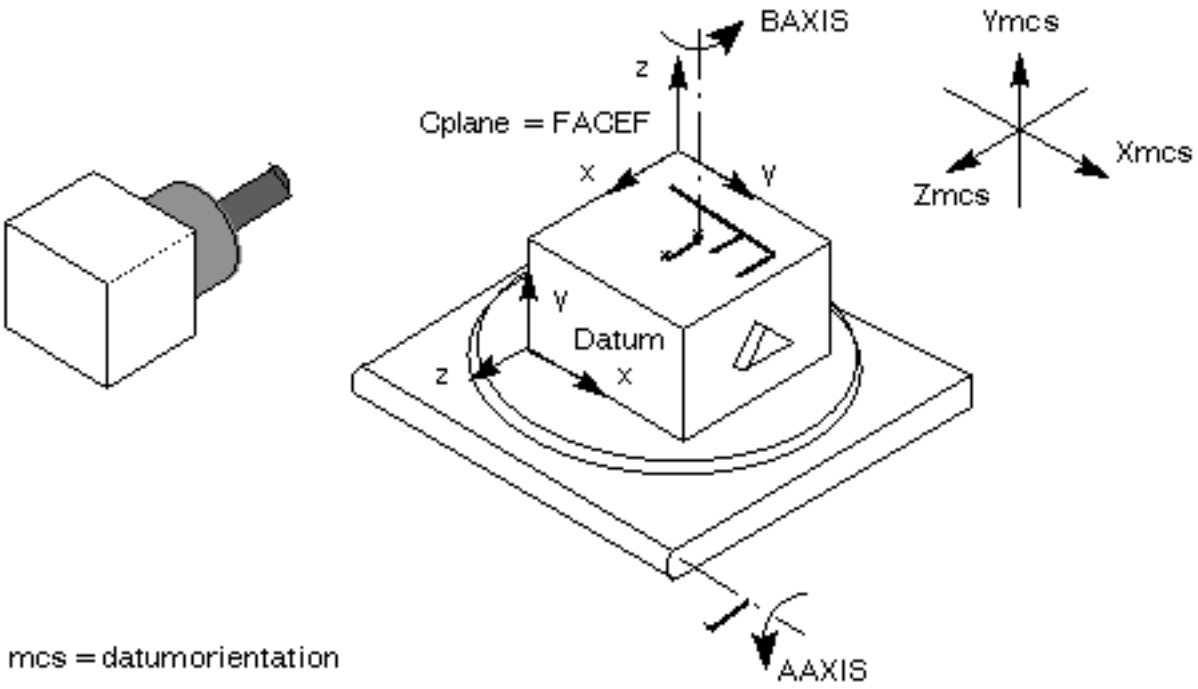
Use the NOOPTIM modifier if condition number one (stated earlier, in the previous section, “Validating INDEX” on page 4-18) is true: the DATUM Cplane can be fully rotated to align with the target Cplane.

The NOOPTIM modifier affects rotation of the machine to exactly reach the specified Cplane. This requires rotating the part and Cplane so that their axes align with the machine coordinate system. If this Cplane cannot be reached exactly, CVNC uses the OPTIM method described in the next section.

NOOPTIM is the system default. Use the DEFINDEX command to change the default to OPTIM.

The following two figures illustrate the use of INDEX with NOOPTIM, showing the initial setup first.

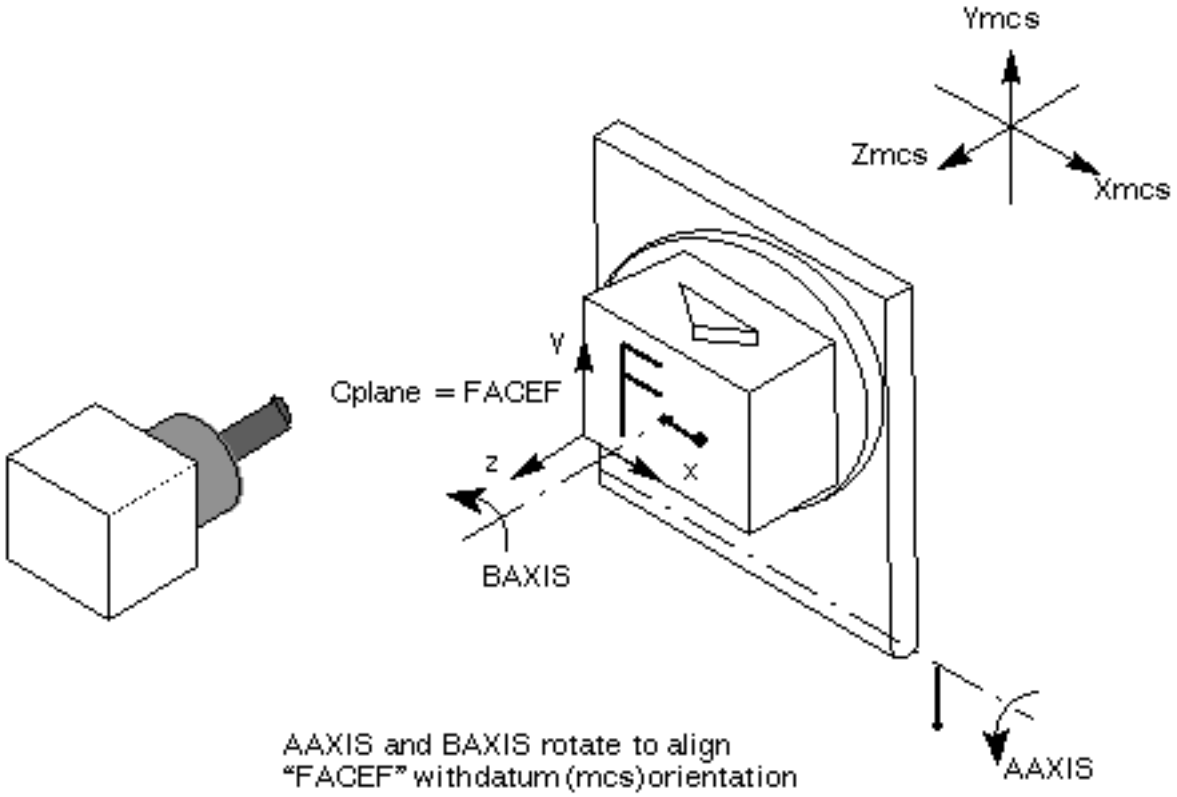
Figure 4-10 Initial Setup: INDEX Used with NOOPTIM



The following example shows the results of executing

```
NC:> INDEX "FACEF" NOOPTIM
```

Figure 4-11 Result: INDEX Used with NOOPTIM



Specifying Tool Axis Only Normal to the Cplane — OPTIM Modifier

Use the INDEX OPTIM modifier to align the tool axis with the z-axis only of the target Cplane.

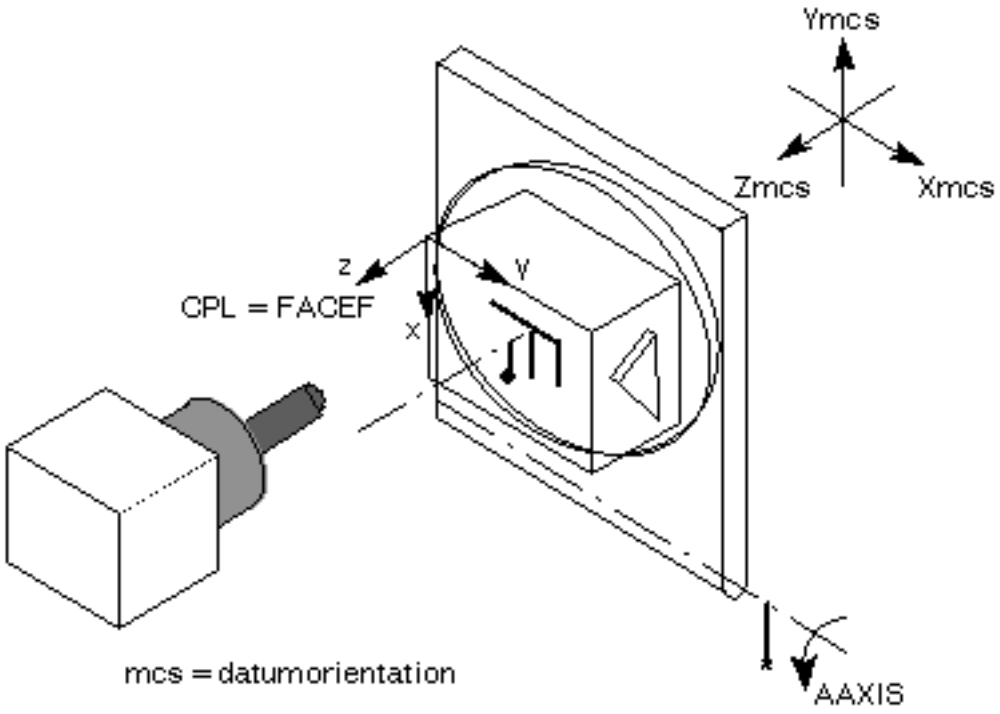
The OPTIM modifier rotates the machine so that the tool axis is normal to the specified Cplane. With OPTIM, the target Cplane x- and y-axes need not be aligned with those of the machine tool. OPTIM aligns the z-axis only and does not try to align the other axes.

To machine the "F" face in the next example, the tool need only be normal to the face. This requires rotation of the a-axis only; b-axis rotation is not necessary.

Thus, if OPTIM is used for the operation shown in the previous figure, the result would be as shown in the following figure.

```
NC:> INDEX "FACEF" OPTIM
```

Figure 4-12 INDEX “FACEF” OPTIM



CLFile ORIGIN Statement Calculation-Rotary Tables

Calculations for ORIGIN, which specifies the machine tool coordinate system origin, are performed automatically.

The calculations described in this section are performed automatically by CVNC and furnished for your information only.

The ORIGIN statement specifies the machine tool coordinate system origin in terms of the part reference system. This is applicable only when table rotation is programmed.

The initial zero location of the part program is established in CVNC through the DATUM command.

When a table is programmed to rotate, the part (plus all other elements mounted on it) reorients and repositions within the machine tool coordinate system.

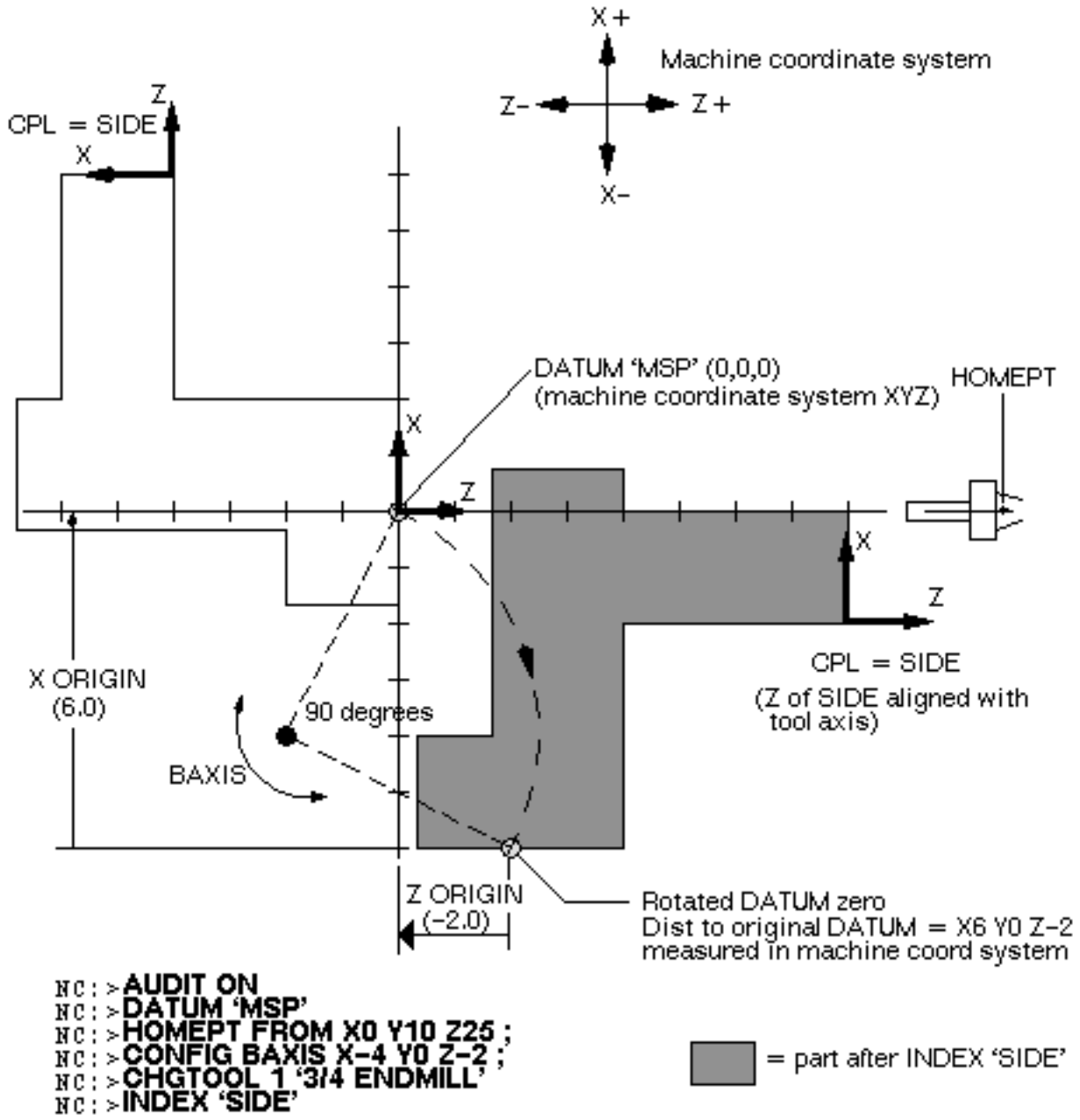
The new position, determined by rotating DATUM X0Y0Z0, is described to the output file through the ORIGIN record in terms of x, y and z measured along the

machine tool axes. The term “rotated datum space” is used in CVNC to describe this new position.

Rotary heads do not affect part repositioning; hence an ORIGIN record need not be generated.

An ORIGIN record is calculated when a 90° rotation of a b-axis table milling machine occurs, as shown in the following figure.

Figure 4-13 Calculation of the ORIGIN Record for a BAXIS Rotary Table



The ORIGIN record generated in the CLFile as a result of the INDEX command is as follows:

```
ORIGIN 6.0000, 0.0000, -2.0000
```

An extract of the CLFile applicable to the previous figure appears in the following table.

Table 4-1 CLFile Applicable to Figure 4-13.

CLFile	Description
PPRINT AUDIT ON PPRINT DATUM 'MSP'	Define initial set up X0Y0Z0 & Machine coordinate system.
PPRINT HOMEPT FROM X0 Y10 Z25 ; FROM 0.0000, 10.0000, 25.0000 PPRINT CONFIG BAXIS X-4 Y0 Z-2 ; PPRINT CHGTOOL 1 '3/4 ENDMILL' LOADTL 1 LENGTH 3.0000 OSETNO 1 PPRINT INDEX 'SIDE'	CONFIGure BAXIS table and pivot point Gage length = 3: #CURLOC => X0 Y10 Z22 Tool change - gage length = 3 Rotate table to work in ACS 'SIDE' - input coordinate system.
ROTATE BAXIS ATANGL 90.0000 CCLW ORIGIN 6.0000, 0.0000, -2.0000	Coordinates from rotated DATUM to part program zero, measured along machine's xyz axes.
PPRINT MOVE XYZLOC X1.44 Y3.75 Z.5 RAPID	Input is with respect to the Cplane 'SIDE'.
GOTO 5.4400, 3.7500, 6.5000	Output is with respect to 'rotated DATUM zero'.
PPRINT MOVE HOME RAPID	Fixed position in machine coordinate system.
GOTO 6.0000, 10.0000, 20.0000	HOME is mapped with respect to rotated DATUM: tool tip position to return head to original location as defined in the HOMEPT command.

The postprocessor converts a CLFile to CNC code using this function:

CNC coordinates = GOTO coordinate - ORIGIN

Thus the CNC code for MOVE `xyzloc` x1.44 y3.75 z.5, using FANUC style code, is:

```
N0200 G00 X-0.56 Y3.75 Z8.5;
```

Machine moves with respect to 'MSP' zero location

Alternatives when Indexing to a Cplane for a 5-Axis Machine

Indexing to a Cplane after configuring a 5-axis machine produces two alternative solutions for reaching the desired destination.

If you configure a 5-axis milling machine and then use INDEX to specify a Cplane to determine the destination orientation, you always have two possible solutions for the move.

Since axis limits are not supported, both alternatives are valid. Both solutions are calculated by CVNC and are displayed in the JCF window for information.

Based upon minimal rotation necessary to achieve the programmed orientation, CVNC selects one of the alternatives and executes the INDEX command.

CVNC selects the solution with the minimal rotation necessary to achieve the programmed orientation and executes the INDEX command.

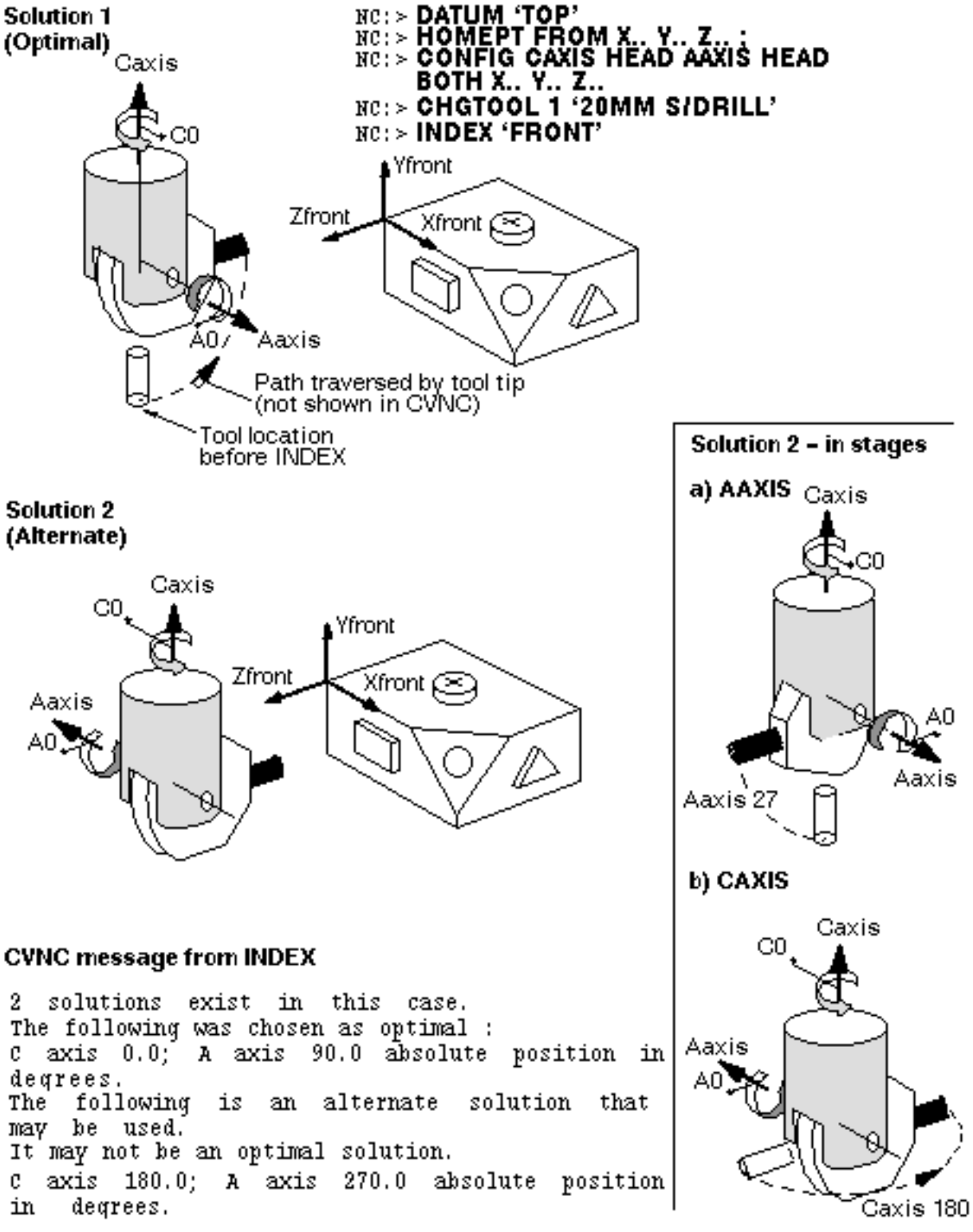
If you prefer the alternative, delete your original INDEX command and replace it with two INDEX commands, using the explicit angle format as supplied in the message generated by the original INDEX command.

Two alternative solutions are given in the following figure. In this figure, the INDEX “FRONT” command generates the alternatives C0 A90 and C180 A270. From the orientation “TOP”, the minimal rotation would be A90°. However, as shown in the two-stage drawing, the same position could be achieved by rotating the AAXIS to 270° and rotating the c-axis through 180°.

If you want the A270 C180 alternative instead of the A90 choice made by CVNC, replace the INDEX “FRONT” command with

```
NC:> INDEX AAXIS 270
NC:> INDEX CAXIS 180
```

Figure 4-14 Two Alternatives for INDEX on a 5-Axis Compound Head Milling Machine



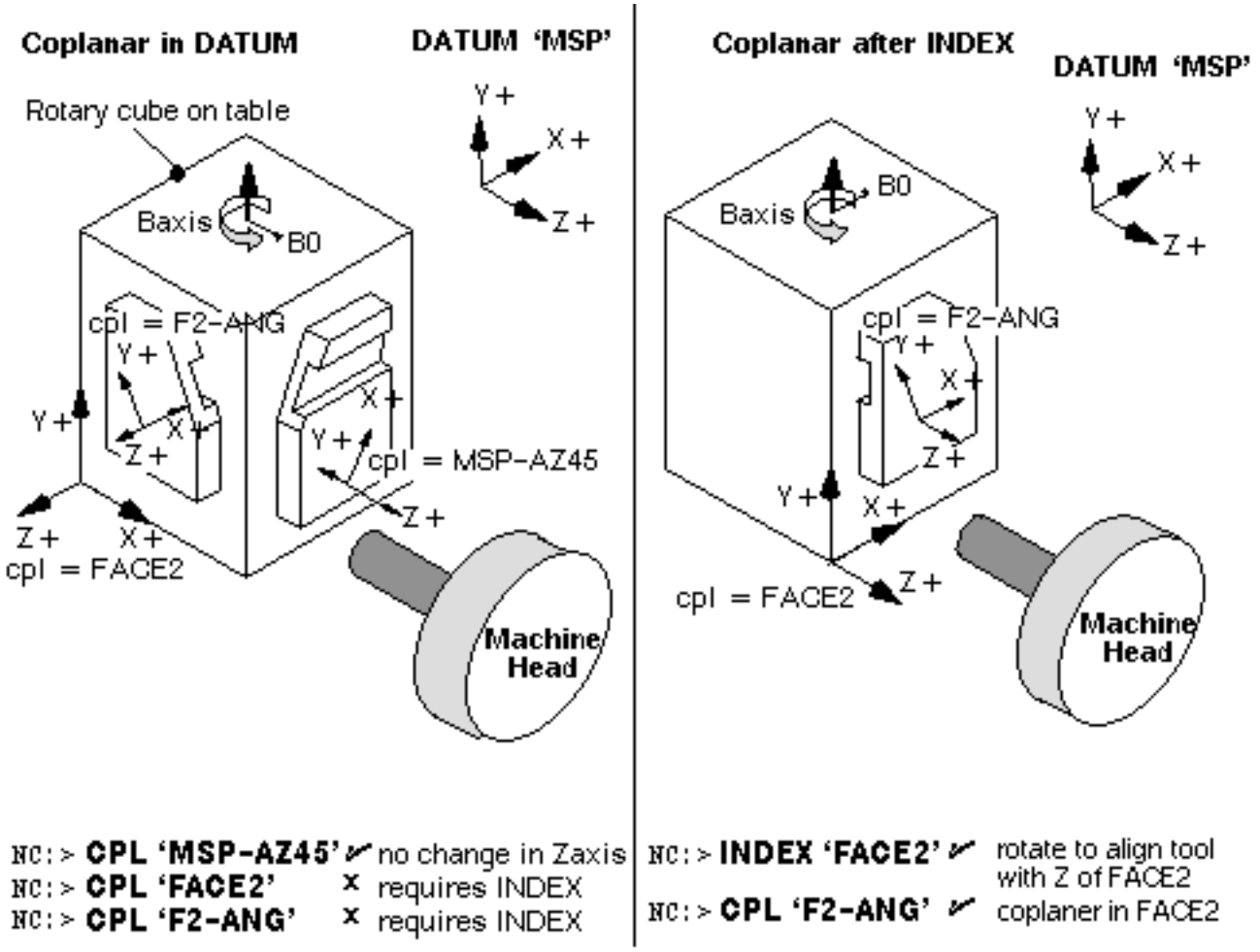
Using the CPL Command with Respect to INDEX

Use the CPL command to select Cplanes in the indexed plane.

The validity of Cplane selection is with respect to the z-axis of the ACS (active Cplane). Thus if the target machine tool has one or more rotary axes, you can use INDEX to reorient the tool axis with respect to the part.

You can select Cplanes in the indexed plane with CPL. The following figure shows an example of how to apply commands to a b-axis horizontal machine tool.

Figure 4-15 CPL Command Validity in Indexed Orientation



You can see from the figure that you can select the "MSP-AZ45" Cplane with CPL when in the DATUM orientation. However, you can select only the "F2-ANG" Cplane after indexing to FACE2.

Notice also that the “F2-ANG” Cplane could have been selected directly without the need for “FACE2” by using INDEX “F2-ANG” NOOPTIM. In that case, the rules of INDEX would be satisfied if the tool axis could be aligned with the z-axis of the destination Cplane.

The Affect of MULTAX ON

You need to select a Cplane that is not constrained by the current tool axis when you are programming multiaxis surface machining. Since the input parameters are often surface related, the machining operation controls the tool axis.

The constraints described above for CPL (the z-axis must not change) are removed when MULTAX is ON. However, CPL does not drive rotary axes.

Input to Output Coordinate System Relationships

For both rotary head and rotary table machine tools, CVNC resolves the machine tool environmental aspects of the work session, leaving you free to focus on the tool and machining functions of manufacturing.

The relationship between the input coordinate system (ACS), in which the user programs in CVNC, and the output coordinate system, which defines the machine tool coordinate system, is different for heads and tables.

CVNC is able to resolve the input-to-output relationship because it knows the type of device each rotary axis is supporting.

Rotary Head Machine Tool

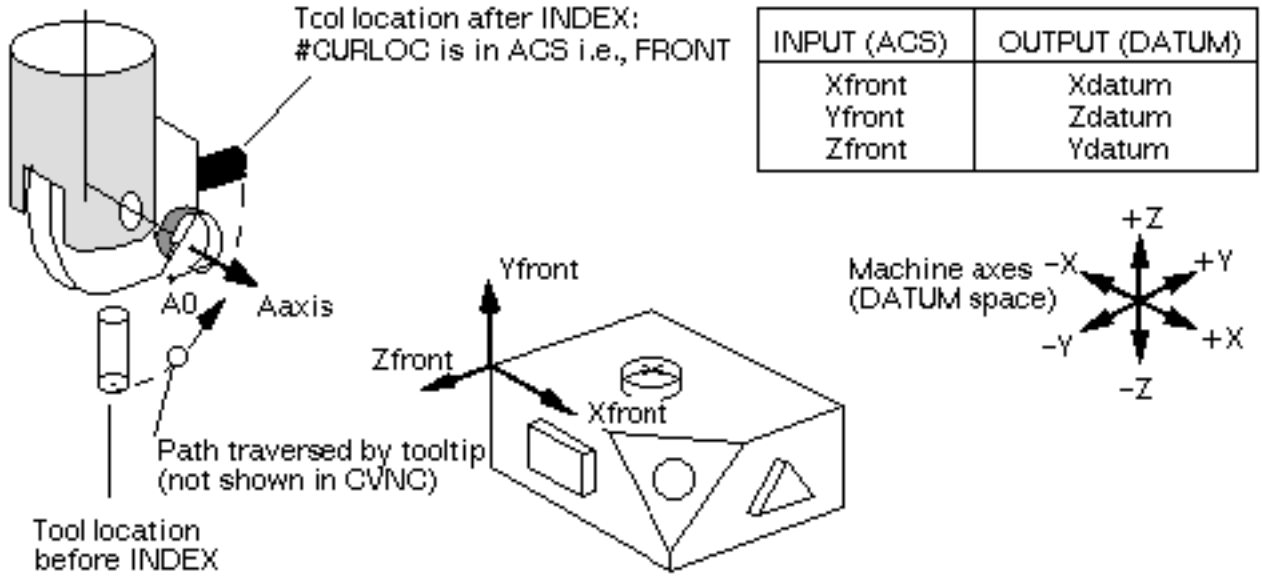
For a rotary head machine tool, you may have entered

```
NC:> CONFIG AAXIS HEAD loc
```

When programmed to rotate, rotary heads change the orientation of the tool with respect to the linear coordinate system of the machine tool.

Consider the input-to-output relationship when an a-axis head machine is indexed through 90°, as shown in the following figure. The Cplane named TOP represents the machine tool coordinate system.

Figure 4-16 Input to Output Relationship: A-Axis Head Machine



INPUT (ACS)	OUTPUT (DATUM)
Xfront	Xdatum
Yfront	Zdatum
Zfront	Ydatum

JCF Command	Action / Results	CLFILE OUTPUT
NC:> **DATUM 'TOP'**	Set machine coord system	None
NC:> **HOMEPT FROM X-100 Y-250 Z300 ;**	Establish gage line locn with respect to DATUM	FROM / -100 -250 300
NC:> **CONFIG AAXIS HEAD X-100 Y-250 Z400 ;**	Define machine rotary axis CONFIGuration : fixed pivot distance is 400	None
NC:> **CHGTOOL 1 '20MM SLOT DRILL'**	Load tool : gage length = 100 Tooltip to pivot axes = 200 in Z	None
NC:> **SHOW #CURLOC; => X-100 Y-250 Z200**	HOMEPT - #TLGAGE : tool aligned with spindle axis	None
NC:> **INDEX 'FRONT'**	The AAXIS rotates 90 degrees ACS - FRONT is selected	ROTATE AAXIS 90 CCLW
NC:> **SHOW #CURLOC ; = > X-100 Y400 Z50**	#CURLOC is with respect to FRONT	None
NC:> **MOVE XYZLOC X10 Y20 Z30 ;**	XYZ in ACS - FRONT OUTPUT is in DATUM	RAPID / GOTO / 10.0, -30.0, 20.0

In the previous figure, the tool is reoriented so that it is normal to the xy-plane of Cplane FRONT. Cplane FRONT is affected by rotation of the a-axis by 90° (INDEX "FRONT").

CVNC must now map the input coordinate system (FRONT) to the machine tool coordinate system (TOP) when output is generated. Thus when a z-axis move is programmed in the JCF, the machine must move its y-axis lead screw. This relationship between input and output axes is shown in the previous figure, on the top right-hand corner.

Rotary Table Machine Tool

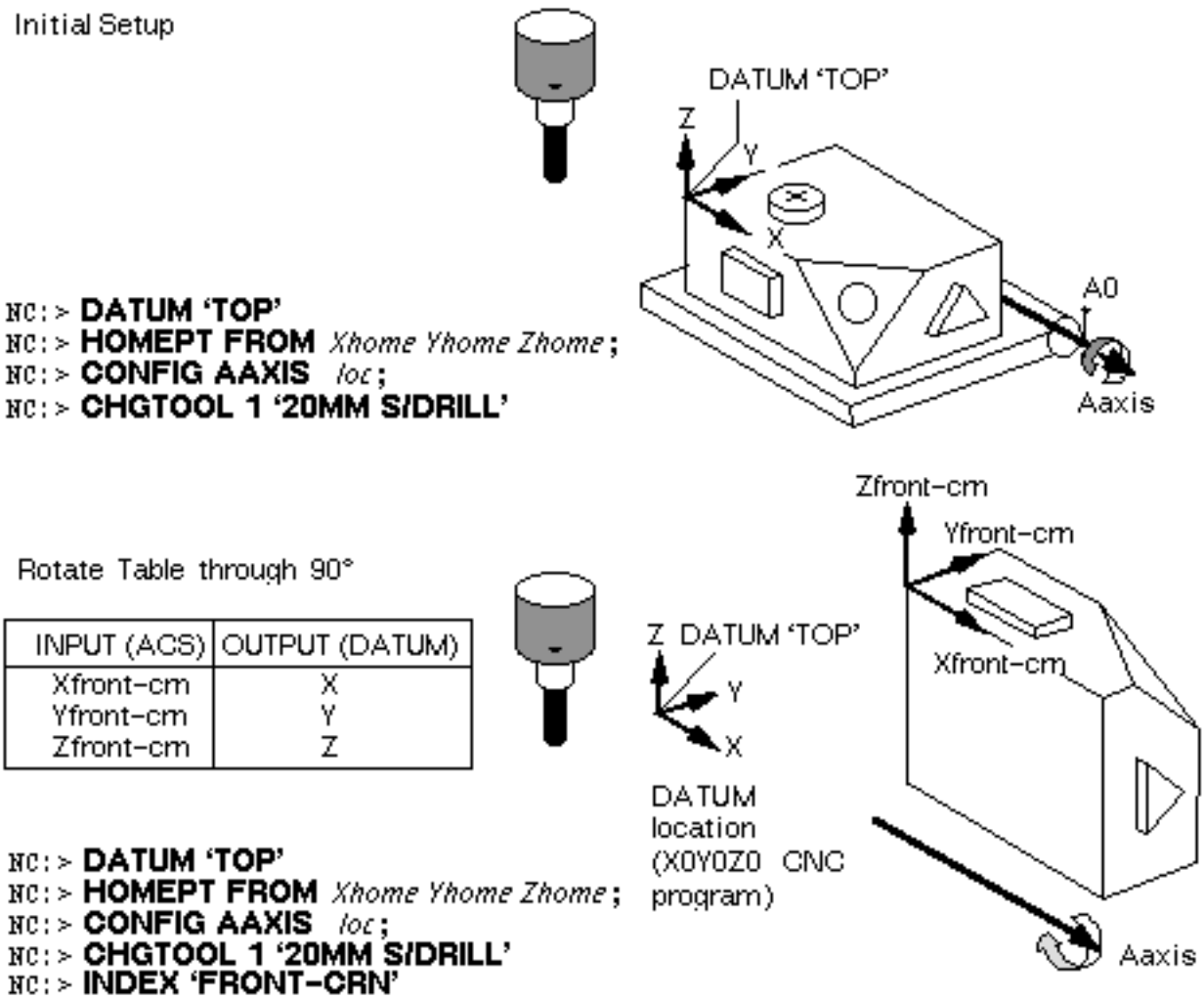
For a rotary table machine tool, you may have entered

```
NC:> CONFIG AAXIS loc
```

When you rotate a table, the part moves and reorients with respect to the machine tool coordinate system. CVNC, using the information defined in CONFIG, resolves the new position of the part to the CNC program location (DATUM).

The next figure shows how the relationship of the part to CNC program zero is determined after a rotation.

Figure 4-17 Relationship of Part to Program Zero: AAXIS Machine



CVNC to NC Machine Relationships

The following examples illustrate the CVNC to NC machine relationship. See page 4-8 for the rules of rotary motion.

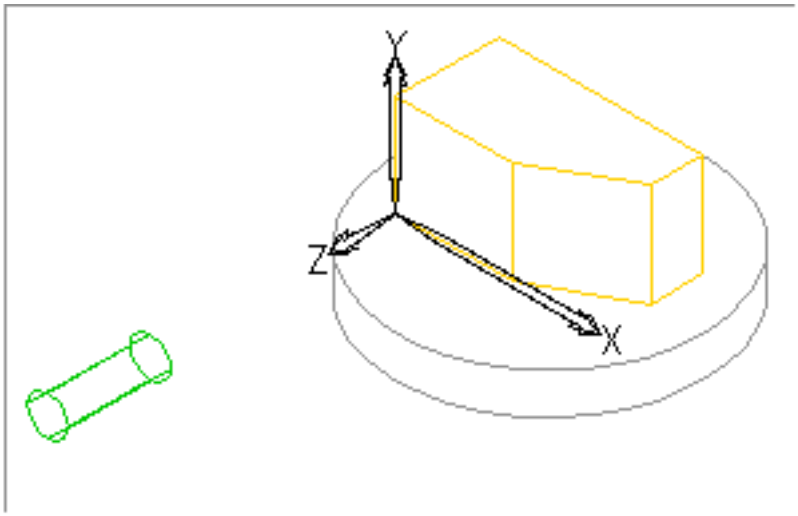
BAXIS Table Device

You can specify the following command to configure your device.

```
NC:> CONFIG AAXIS PRIMARY BAXIS BOTH
```

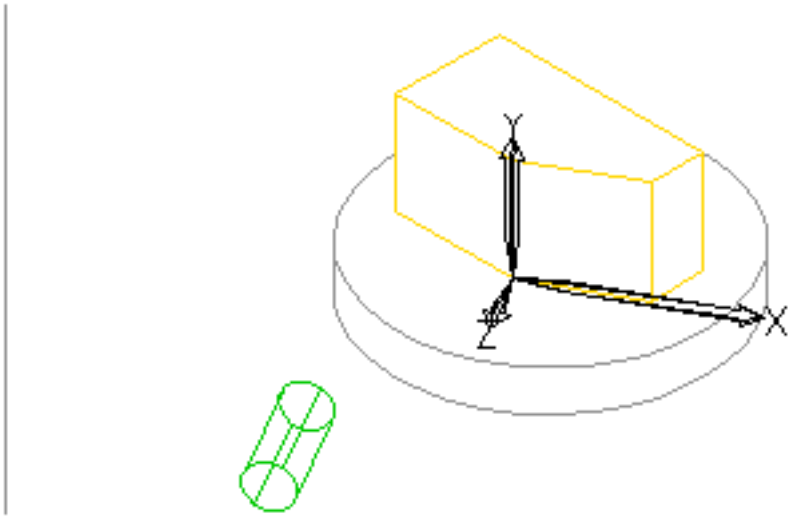
The following figure shows a chamfer with a face oriented at an angle of 30° about the y-axis.

Figure 4-18 CVNC Graphical Output before INDEX for table Device



After executing an INDEX command, CVNC positions the tool as shown in the following figure.

Figure 4-19 CVNC Graphical Output after INDEX for Table Device



The following table explains the actions with the JCF and CLFile output.

Table 4-2 JCF and CLFile Output for Table Device

JCF Command	Action/Result	CLFile Output
NC:> DATUM 'Front'	Set machine coordinate system	None
NC:> INDEX 'BAX+30'	Index to the defined Cplane attached to the face (BAX+30 is a CPL defined offset from DATUM, oriented at 30° about the y-axis.)	ROTATE BAXIS ATANGL 30 CCLW

The two figures below show the actual positioning on the NC machine. This illustrates the difference between the output in CVNC and the actual output on the NC machine.

Figure 4-20 NC Machine before INDEX for Table Device

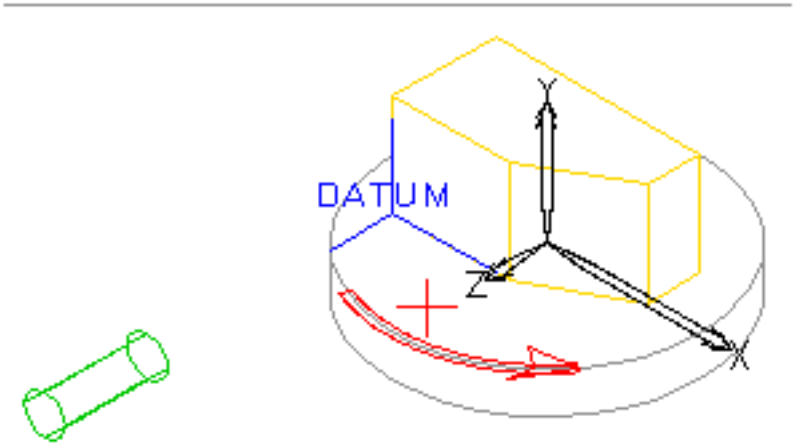
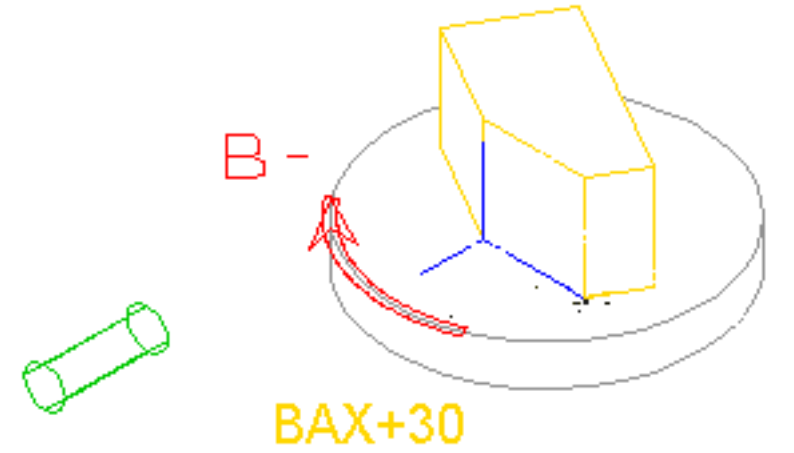


Figure 4-21 NC Machine after INDEX for Table Device



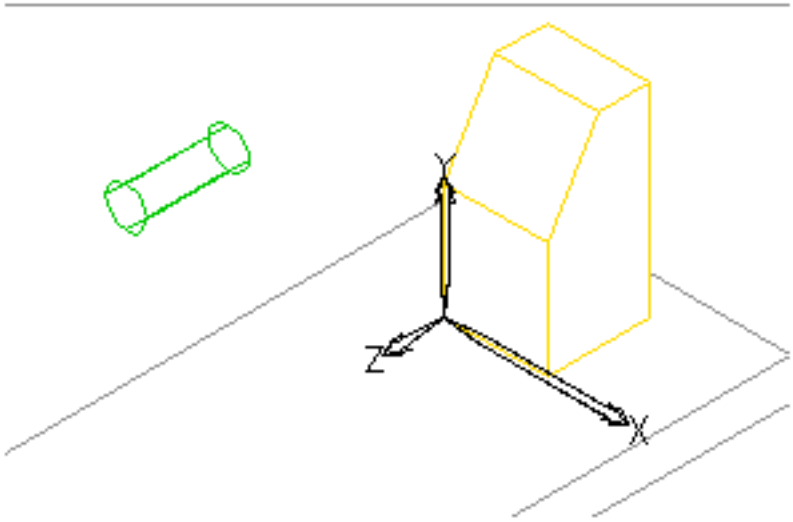
AAXIS Head Device

You can specify the following command to configure your device.

```
NC:> CONFIG AAXIS HEAD BAXIS BOTH
```

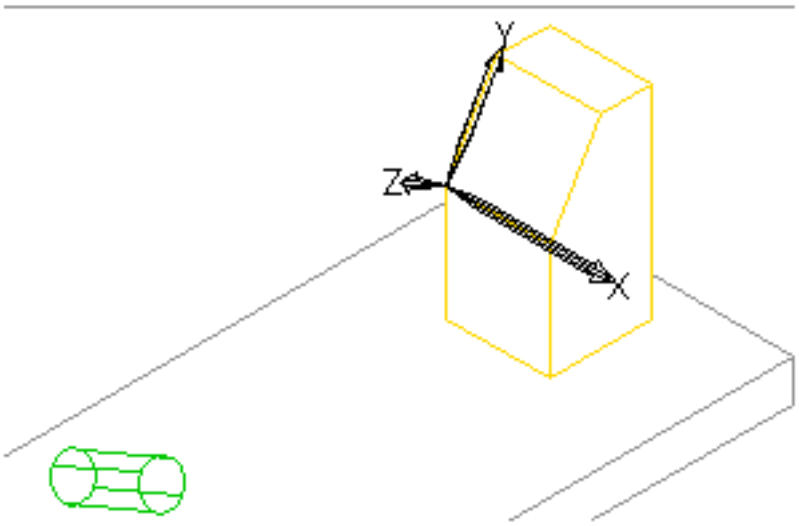
The following figure shows a chamfer with a face oriented at an angle of -30° about the y-axis.

Figure 4-22 CVNC Graphical Output before INDEX for Head Device



After executing an INDEX command, CVNC positions the tool as shown in the following figure.

Figure 4-23 CVNC Graphical Output after INDEX for Head Device



The following table explains the actions with the JCF and CLFile output.

Table 4-3 JCF and CLFile Output for Head Device

JCF Command	Action/Result	CLFile Output
NC:> DATUM 'Front'	Set machine coordinate system	None

Table 4-3 JCF and CLFile Output for Head Device

JCF Command	Action/Result	CLFile Output
NC:> INDEX 'AAX-30'	Index to the defined Cplane attached to the face (AAX-30 is a CPL defined offset from DATUM, oriented at -30° about the x-axis.)	ROTATE AAXIS ATANGL 330 CLW

Please note: In CVNC output, the angle is always an absolute value; it cannot be negative.

The two figures below show the actual positioning on the NC machine. This will illustrate the difference between the output in CVNC and the actual output on the NC machine.

Figure 4-24 NC Machine before INDEX for Head Device

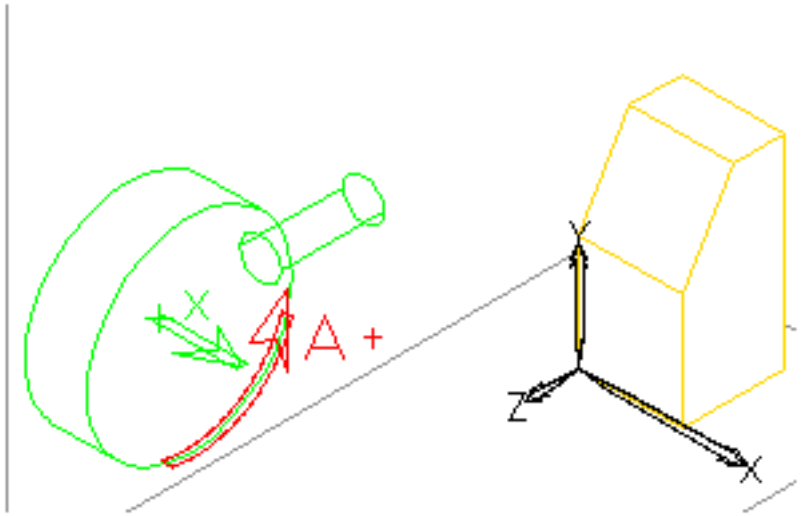
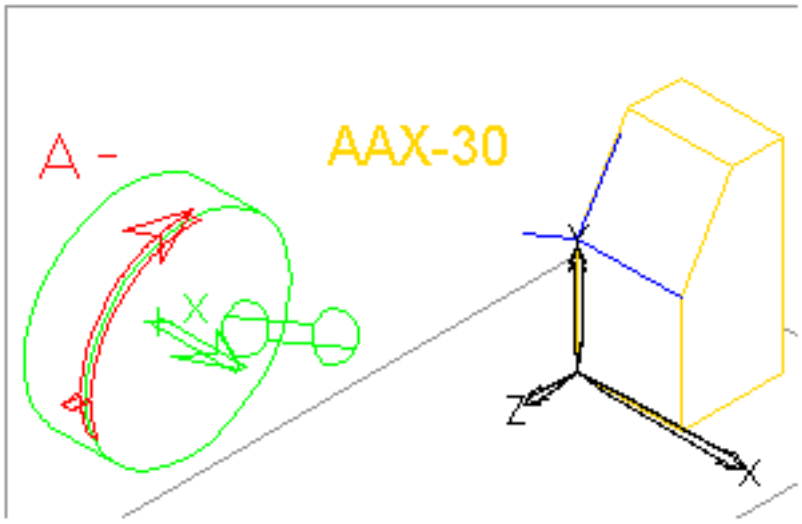


Figure 4-25 NC Machine after INDEX for Head Device



Chapter 5

Operation Setup: Operational Parameters

The commands described in this chapter are used to set a number of operational parameters for your operation, such as reference planes, speed and feed rates, coolant flow, diameter compensation register, and stock and tolerance values.

- Overview of Setting Operational Parameters
- Setting Z-Planes — PLANE
- Specifying Modal Feed Rates — FEED
- Specifying Spindle Speed — SPEED
- Turning Coolant On and Off — COOLANT
- Setting Stock Offsets — STOCK
- Specifying Tolerance — TOLER
- Setting the Diameter Compensation Register — DIACOMP
- Calculating Tool Offset — CALCRAD
- Multiaxis Output — MULTAX

Overview of Setting Operational Parameters

This section gives an overview of the operational parameters.

Reference Planes

Before beginning a milling or hole processing operation, use PLANE to set up the z-planes your tool will reference as it makes cutting and noncutting movements.

A z-plane is perpendicular to the tool axis, located a specified distance from the active Cplane. Z-planes are used as a reference point for cutting, plunging, approaching, clearing, and retracting.

Speed and Feed Rates

For each milling or hole processing operation, you must specify the speed at which the tool rotates and the feed rate at which the tool moves.

Use SPEED to specify the speed at which the tool will rotate. You can specify speed in revolutions per minute (RPM), surface feet per minute (SFM), or surface meters per minute (SMM). The default is zero RPM.

Use FEED to specify the feed rate at which the tool moves during cutting and noncutting motion.

Coolant

You will want to use the COOLANT command when cutting metal. An alternative to the COOLANT command is the TLCHG macro, which performs tool changing tasks as well as turning the coolant on and off.

Stock and Tolerance Values

The STOCK command allows you to leave material on your part, at varying amounts along different entities, for the finish cut. You can choose how fillets are machined when they are contiguous to entities with varying amounts of stock.

You can specify a tolerance zone at both sides of a circular cutting path with the TOLER command. CALCRAD calculates the tool offset necessary to cut an entity when the entity height is equal to the tool corner radius.

Diameter Compensation Register

With DIACOMP, you can set the diameter compensation register on your machine tool to either the left or right of the tool path to compensate for factors such as tool wear or different tool sizes.

Setting Z-Planes — PLANE

Set up reference, approach, machining, retraction, and clearance planes relative to your active Cplane.

A z-plane is perpendicular to the tool axis, located a specified distance from the active Cplane. Z-planes are used as a reference point for cutting, plunging, approaching, clearing, and retracting.

The z-planes available are

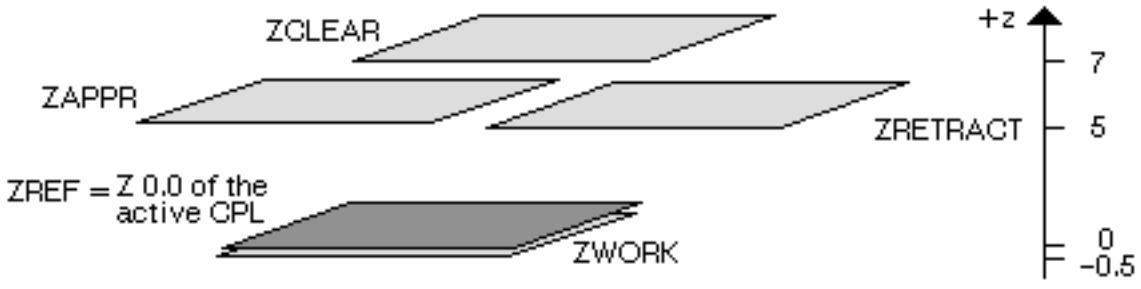
- ZREF (reference)
- ZAPPR (approach)
- ZWORK (machining)
- ZRETRACT (retraction)
- ZCLEAR (clearance)

Before beginning a milling or hole processing operation, use PLANE to set up the z-planes your tool will reference as it makes cutting and noncutting movements.

For example, to set the z-planes shown in the following figure, enter the following:

```
NC:> PLANE ZREF 0
NC:> PLANE ZAPPR 5
NC:> PLANE ZWORK -.5
NC:> PLANE ZRETRACT 5
NC:> PLANE ZCLEAR 7>
```

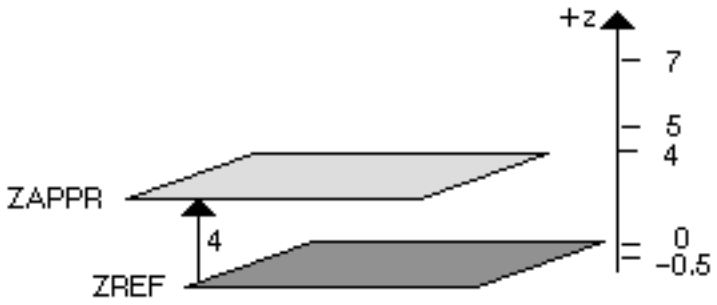
Figure 5-1 Sample CVNC-M2 Z-Planes



The ZREF plane is used as a reference for setting the values of other planes. In the following figure, the ZAPPR is set at an incremental value of 4 inches relative to the location of the ZREF plane. To do this, enter

```
NC:> PLANE ZREF 0
NC:> PLANE ZAPPR 4 ZREF
```

Figure 5-2 Using the ZREF Plane



Z-Plane Defaults

If no z-planes are set, CVNC defaults to the values listed below.

Z-plane	Default Value
ZREF	0.0 inches/0.0 mm
ZAPPR	10.0 inches/255.0 mm
ZCLEAR	15.0 inches/380.0 mm
ZRETRACT	10.0 inches/255.0 mm
ZWORK	0.0 inches/0.0 mm

For ZAPPR, ZCLEAR, and ZRETRACT, if only one of these z-planes is defined, the other two default to that same value. If two are defined, the third is set at the last value entered for one of the other two. For example, if ZCLEAR is set at 20 and ZAPPR is set at 30, ZRETRACT defaults to 30.

Specifying Modal Feed Rates — FEED

Use FEED to specify the modal feed rate of tool motion during the performance of cutting and noncutting motion commands.

FEED specifies modal feed rates for motion commands. Modal feed rates are activated when subsequent motion commands are entered. One cutting or noncutting motion operation can contain several modal feed rates.

For example, a pocketing operation that includes APPROACH, PLUNGE, and CUT tool motion activates APPROACH, PLUNGE, CUT, and CONNECT feed rates.

Please note: CONNECT feed rates apply to AREAMILL and SURFCUT3 LACE (CVNC-M3) connections only.

If a motion command has no default feed rate or if you want to override a default, you must assign a feed rate with a numerical expression.

Feed Rate Defaults

CVNC defaults to the following modal feed rates if you do not define values for these commands with the FEED command.

Table 5-1 Feed Rate Defaults

CVNC Defaults	Modal Feed Rates
APPROACH	RAPID
CUT	0
CUT ARC, CUT ENT, PROFILE, POCKET PLUNGE, CONNECT	CUT feed rate
CLEAR	RETRACT feed rate (if set), otherwise RAPID
RETRACT	CLEAR feed rate (if set), otherwise RAPID

These modal feed rates default as specified until you set them.

For example, to assign a modal cutting feed rate of .005 millimeters per revolution, enter

```
NC:> FEED CUT .005 MMPR
```

Slowdown Feed Rates

A number of tool path generators in CVNC create sharp direction changes in tool motion, resulting in an undesired surface finish. In such cases, you can use the SLOWDN modifier for a better surface finish.

Please note: The SLOWDN modifier for the FEED command, works only with the ZPROF3, SURFCUT3, PROFILE, POCKET, MPOCKET, PROFILE5, SURFINT5, and SWARFCUT tool motion commands.

Sharp Corners

At a distance specified, before a sharp corner, a tool path point is generated, and the cutting feed rate is changed to one specified. This cutting feed rate is the slowdown feed rate.

A sharp corner is defined as any ‘inside’ change in tool motion direction, greater than an angle specified. Slowdowns do not apply to ‘outside’ sharp corners.

The tool then turns the corner at the slowdown feed rate and continues for the same distance past the corner. The original feed rate is then reinstated.

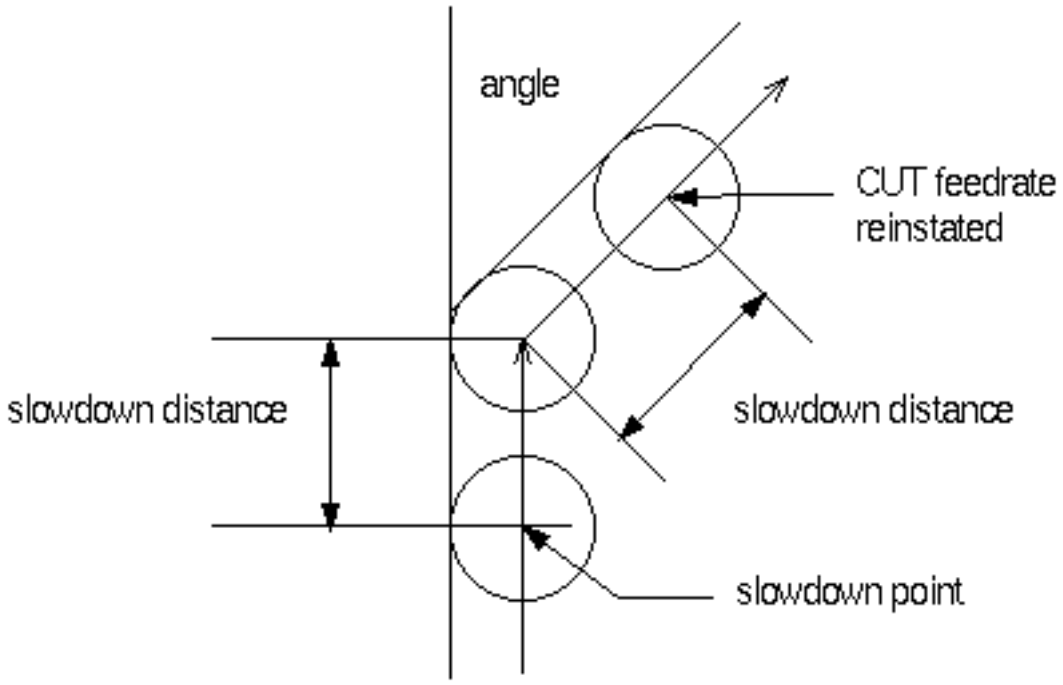
The default slowdown feed rate is the CUT feed rate, that is, no slowdown. The default distance is 1” or 25 mm, while the default slowdown angle is 10 degrees. The slowdown feed rate may be less or greater than the CUT feed rate.

For example,

```
NC:> FEED SLOWDN 8 DIST 0.5 ATANGL 12
NC:> FEED CUT 20
```

This example produces a cutting feed rate of 20 ipm up to a distance of 1/2 an inch from the corner. The feed rate then changes to 8 ipm for the 1/2 inch motion into and out of the corner. The original cutting feed rate of 20 ipm then resumes. It applies to all ‘inside’ direction changes greater than 12 degrees.

Figure 5-3

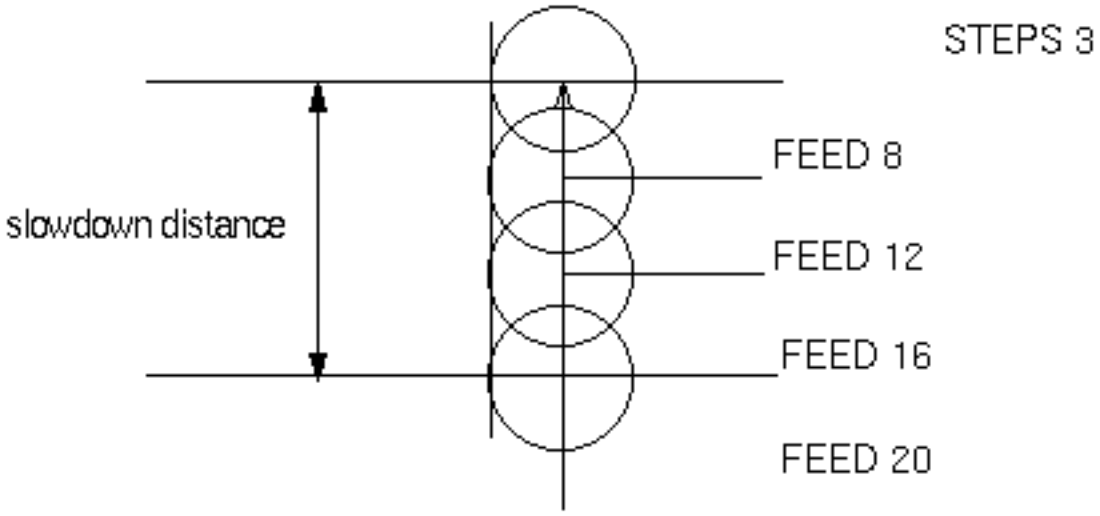


If slowdown distances overlap because the distance between two corners is less than two times the slowdown distance, the tool accelerates up to the midpoint and then slows down again, never reaching the CUT feed rate.

Acceleration and Deceleration

Acceleration and deceleration is carried out by optionally dividing the slowdown distance into a number of steps. The slowdown feed rate is used for the last step, and proportionately decreasing or increasing feed rates are used for the intermediate steps. The default number of steps is 1.

Figure 5-4



For example,

```
NC:> FEED SLOWDOWN 8 DIST 0.5 ATANGL 12 STEPS 3
NC:> FEED CUT 20
```

Arc Motion

Slowdown feed rates also apply as the tool approaches both ‘inside’ and ‘outside’ arcs. The objective is to avoid gouges or machining marks from overruns into the arc, and control feed rates around the outside of small arcs.

At a distance specified, before an arc, a tool path point is generated, and the cutting feed rate is changed to one specified. This cutting feed rate is the slowdown feed rate.

An arc is defined as any circular motion with a tool center radius less than the active tool radius.

The motion around the arc, and for an equal distance after the arc, is at the slowdown feed rate. The original feed rate is then reinstated.

The default slowdown feed rate is the CUT feed rate, that is, no slowdown. The default distance is 1” or 25 mm.

For example,

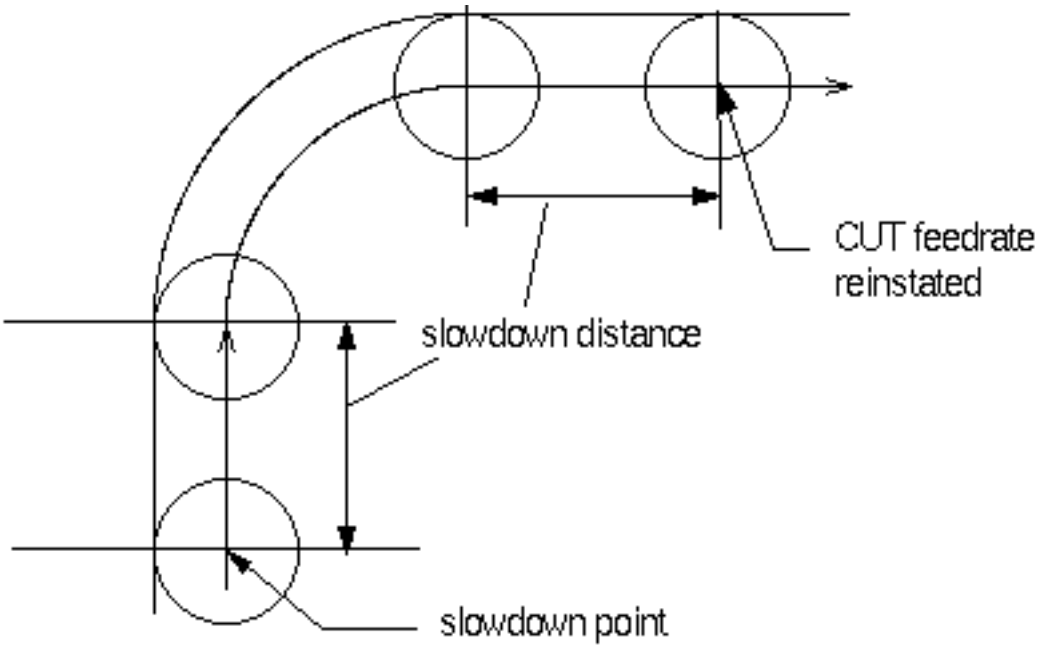
```
NC:> FEED SLOWDOWN 8 DIST 0.5 MRAD 2
```

```
NC:> FEED CUT 20
```

This example produces a cutting feed rate of 20 ipm up to a distance of 1/2 an inch from the arc. The feed rate then changes to 8 ipm for the 1/2 inch motion approaching the arc, around the arc, and 1/2 an inch after the arc. The original cutting feed rate of 20 ipm is then reinstated. It applies to all tool center arcs with a radius less than 2 times the tool radius.

Acceleration and deceleration apply as mentioned earlier.

Figure 5-5



Specifying Spindle Speed — SPEED

Set the speed of the spindle, its direction, and range with the SPEED command.

SPEED specifies the spindle speed for a milling operation. This speed can be specified in one of the following:

- Revolutions per minute
- Surface feet per minute
- Surface meters per minute

This is a modal parameter, so spindle speed need not be specified again after the SPEED command has been entered unless a change is desired.

This command is also used to turn the spindle on and off.

You can specify clockwise rotation for the spindle (the default) or counterclockwise rotation.

You can also specify the spindle range, whether high, medium, low, or a particular value, and the maximum revolutions per minute allowed for the machine tool.

For example, to specify 150 revolutions per minute counterclockwise at a medium range, enter

```
NC:> SPEED RPM 150 CCLW RANGE MEDIUM
```

Turning Coolant On and Off — COOLANT

The COOLANT command turns the coolant off and on or sets it to a different flow rate.

Use COOLANT to turn the coolant on and off or to adjust its volume.

You can set COOLANT at:

- ON (normal flow)
- OFF (the default)
- FLOOD (heavy flow)
- MIST (fine, light flow)

For example, to turn on a heavy flow of coolant, enter

```
NC:> COOLANT FLOOD
```

Setting Stock Offsets — STOCK

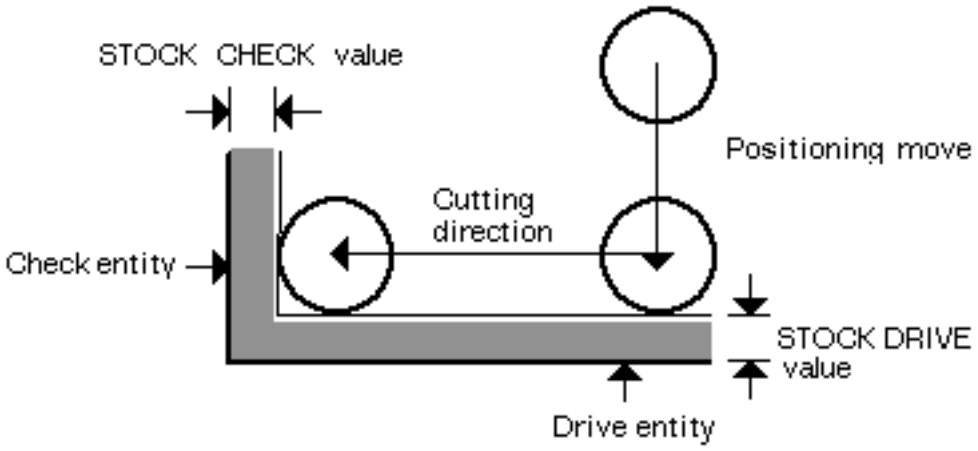
The STOCK command allows you to add thickness to an entity or boundary. Thickness can vary along different entities.

You can affect the behavior of motion commands by setting offsets with STOCK.

STOCK allows you to add thickness to an entity or boundary. STOCK can vary along different entities being machined by the same command.

STOCK can be added to a drive entity and a check entity. A drive entity is an entity that the tool moves TO/ON/PAST or along in a cutting operation. A check entity is an entity that a tool moves TO, ON, or PAST in a checking operation. The figure given below shows how the STOCK command can be applied to each.

Figure 5-6 STOCK Values



STOCK offsets added to drive entities with STOCK DRIVE are used in the following operations:

- AREAMILL
- PROFILE
- POCKET
- CUT (TO/ON/PAST)
- CUT ENTITY

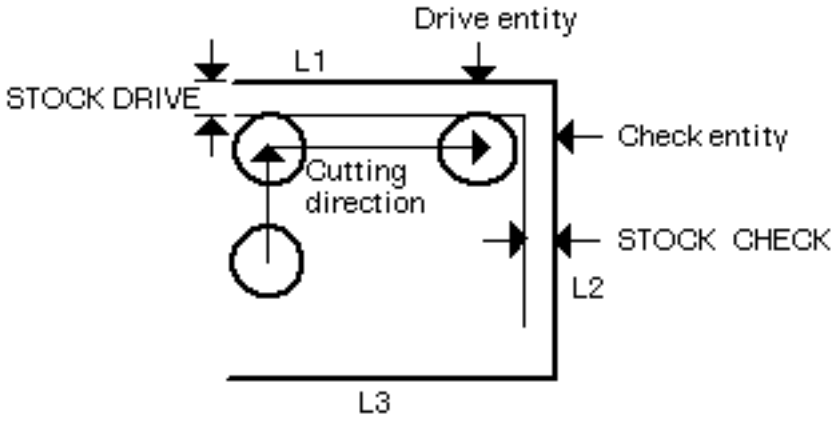
STOCK offsets added to check entities with STOCK CHECK are used in CUT ENTITY operations. CUT ENTITY uses the STOCK DRIVE value for the entity

being machined along and the STOCK CHECK value to CUT TO or PAST the next entity.

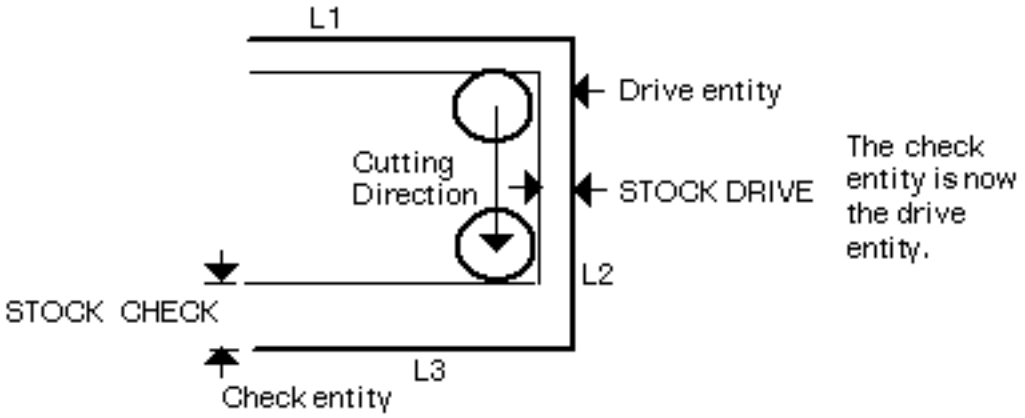
After each CUT ENTITY command, the STOCK CHECK becomes the STOCK DRIVE value. To change STOCK CHECK between CUT ENTITY commands, follow the examples given below.

Figure 5-7 Replacing STOCK CHECK with STOCK DRIVE

```
NC:> STOCK DRIVE .25
NC:> STOCK CHECK .25
NC:> CUT TO $L1;
NC:> CUT CHECK TO $L2;
```



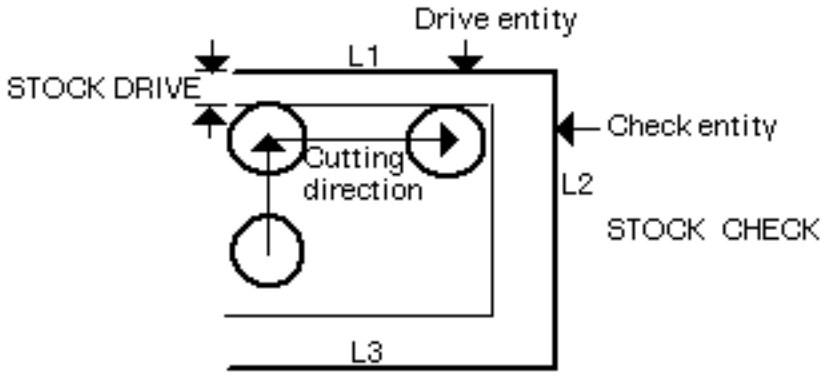

```
NC:> STOCK CHECK .50
NC:> CUT CHECK TO $L3;
```



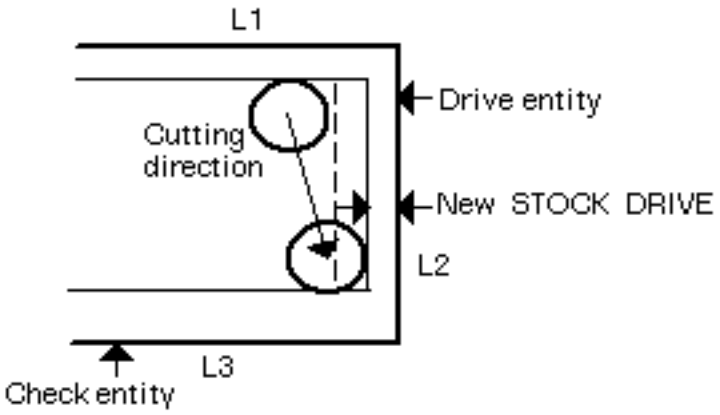
You can set STOCK DRIVE or change STOCK CHECK between CUT ENTITY commands. To change STOCK DRIVE between CUT ENTITY commands, resulting in an angular move, follow the examples in the next figure.

Figure 5-8 Setting STOCK DRIVE Between CUTs

```
NC:> STOCK DRIVE .25
NC:> STOCK CHECK .50
NC:> CUT TO $L 1;
NC:> CUT CHECK TO $L2
```



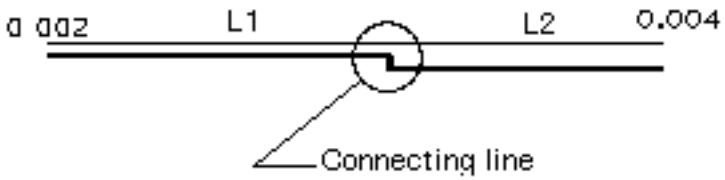

```
NC:> STOCK DRIVE .25
NC:> CUT CHECK TO $L3;
```



If the stock values of tangent entities are not the same, STOCK offsets entities and connects the end points with a straight line as shown in the following figure.

```
NC:> STOCK .0.002 $L 1;0.004 $L2
NC:> PROFILE TO $L 1 $L2;ZYSIDE ....
```

Figure 5-9 Entities with Unequal Stock Values

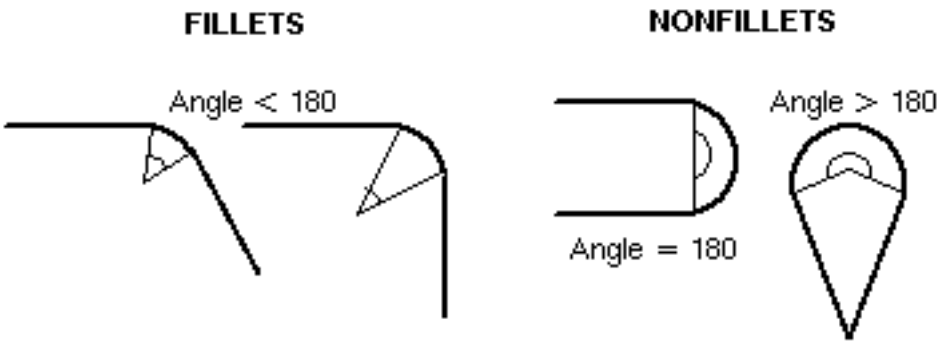


Specifying Fillet Treatment

Use STOCK command modifiers to specify how fillets are to be machined. The fillet radius arc can be floated or fixed.

Use STOCK to specify stock for individual entities.

STOCK can specify how fillets are treated when adjacent entities have stock assigned. In this case, a fillet is defined as an arc less than 180°and tangent on both ends to its adjacent entities.



The ARCFLOAT and ARCFIX modifiers are used with STOCK to specify how machining behaves along boundaries that have fillets.

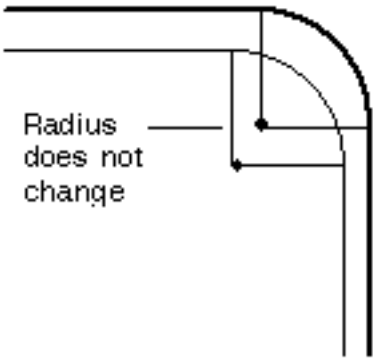
ARCFLOAT floats fillets without a stock value. This maintains tangency between the fillet and the rest of the boundary.

ARCFIX specifies that a fillet remain fixed (not floated).

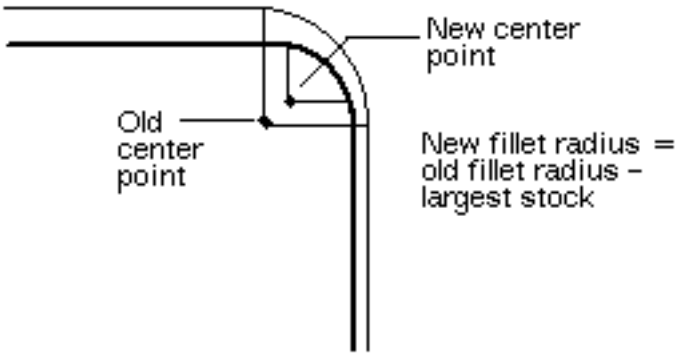
Conditions when a Fillet Is Floated

A fillet is floated under two conditions:

- ARCFLOAT is selected and the fillet has no stock value.
- If machining is done on the outside of a fillet, the arc is floated so that the radius remains the same and the center changes. If machining is done on the inside of a fillet, the radius is reduced by the larger of the two stock values applied to the adjacent entities, and a new center is created. (See the following diagrams.)

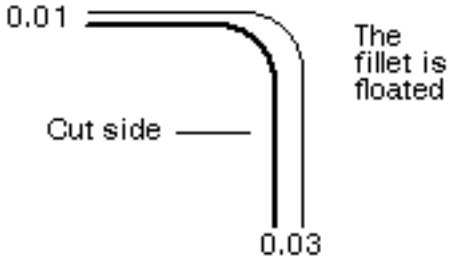


Machining on outside of fillet



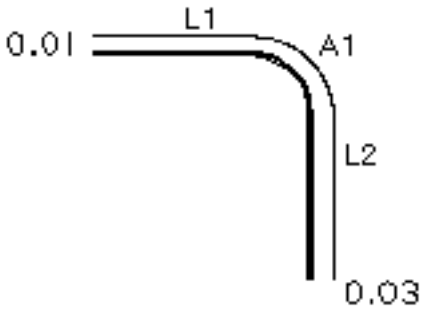
Machining on inside of fillet

Stock for entities adjacent to a fillet are applied to the same side as the cut.



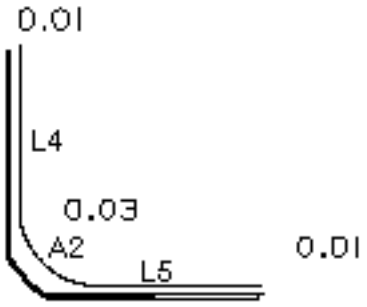
For example, if you have a fillet with no stock value and you want to float it to maintain tangency at both ends, enter

```
NC:> STOCK 0.01 $L1; 0.03 $L2;
NC:> STOCK ARCFLOAT
NC:> PROFILE TO $L1 $A1 $L2; XYSIDE X1.0 Y1.0;
```



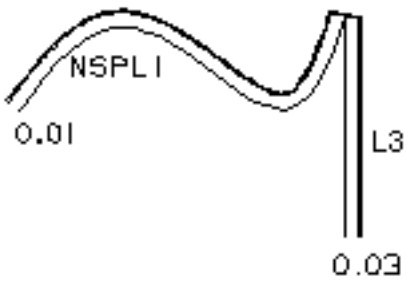
If entities intersect when the fillet is offset, all entities are offset.

```
NC:> STOCK 0.01 $L4 $L5;0.03 $A2;
NC:> PROFILE TO $L4 $A2 $L5; XYSIDE X1.0 Y1.0;
```



If entities do not intersect when the fillet is offset, end points are connected with a straight line.

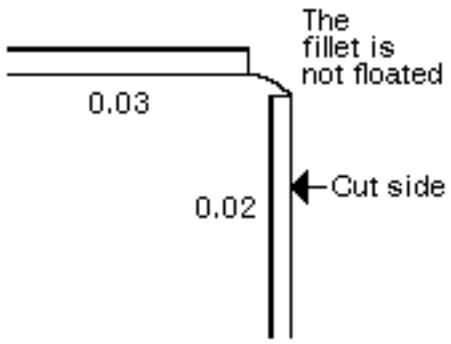
```
NC:> STOCK 0.02 $NSPLI $L3; XYSIDE X1.0 Y1.0;
NC:> PROFILE TO $NSPLI $L3; XYSIDE X1.0 Y1.0;
```



Conditions when a Fillet Is Not Floated

A fillet is not floated when

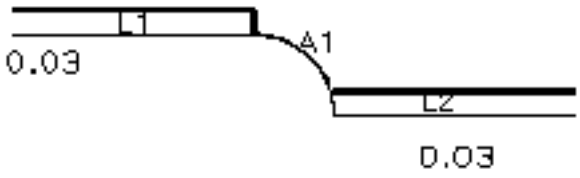
- ARCFLOAT is selected and the fillet has stock. The fillet is offset by the specified stock value.
- ARCFIX is selected.
- Stock for entities adjacent to the fillet is not applied to the same side.



For example, if you have fillets that are only tangent at one end and they are not floated, enter

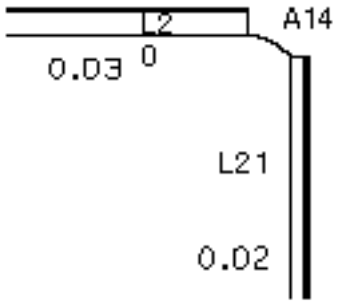
```
NC:> STOCK .03 $L1 $L2;
NC:> PROFILE TO $L1 $A1 $L2; XYSIDE....
```

This produces the following results.



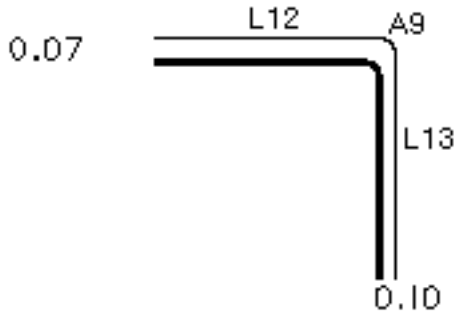
If fillets are not tangent, they are not floated.

```
NC:> STOCK ARCFLOAT
NC:> STOCK .03 $L20; .02 $L21;
NC:> PROFILE TO $L20 $A14; XYSIDE...
```



If the offset of a fillet is toward the inside of the fillet and greater than the radius, then the fillet is trimmed.

```
NC:> STOCK ARCFLOAT
NC:> STOCK .07 $L12 .10 $L13;
NC:> PROFILE TO $L13 $A9 $L12; XYSIDE...
```



Specifying Tolerance — TOLER

TOLER sets the inner and outer tolerance values for machining motion on a boundary or surface.

Use the INTOL modifier to specify the interior tolerance of the tool path. The interior tolerance is the amount the tool is allowed to cut into (beyond) the surface or boundary material at any point.

Use the OUTTOL modifier to specify the exterior tolerance. This is the amount of material allowed outside the surface or boundary, measured normal to the surface or boundary.

For example, to specify an interior tolerance zone of .005 and an exterior tolerance of .0025, enter

```
NC:> TOLER INTOL .005 OUTTOL .0025
```

Please note: CVNC displays circular moves as a series of point-to-point moves, which stay within the INTOL and OUTTOL zones. This motion is written as circular interpolation records and as point-to-point data within a CLFile. Circular moves could be derived from circles, arcs, sections of curves, or from the CUT ARC command.

Postprocessors for machine tools that support circular interpolation do not usually use this coordinate data for driving the circular motion of the machine.

Setting the Diameter Compensation Register — DIACOMP

DIACOMP sets the cutting tool diameter compensation to either the right or left of the tool path. You must specify the amount of compensation.

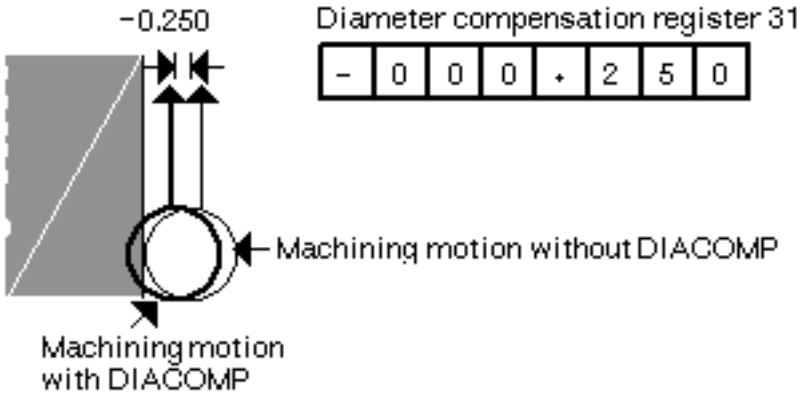
Use DIACOMP to set the diameter compensation register on your machine tool to either the left or right of the tool path. This allows you to offset tool motion from the coordinates specified in the JCF to compensate for factors such as tool wear or different tool sizes.

Each machine tool has its own register, a buffer or storage location.

The following figure shows an example of how diameter compensation affects the tool-to-part relationship. For example, to set the diameter compensation to the right of your tool path, using the value in register 31, enter

```
NC:> DIACOMP 31 RIGHT
```

Figure 5-10 Diameter Compensation



Contact Point Output

Use the NORMAL option to control the generation of contact point output, that is, the output of the contact point of the tool and the entity being machined, and the surface normal of the entity at that point, along the tool axis. This enables you to give cutter compensation in multi-axis mode including 3-, 4-, and 5-axis tool path motion.

DIACOMP NORMAL generates CUTCOM NORMDS in the CLFile and APTsource files. This output is generated only if MULTAX is turned on and DIACOMP NORMAL is selected before the surface machining tool path is created.

The output record is made up of three components:

Table 5-2

XYZ	Contact point co-ordinates of the tool with the entity being machined
IJK	The surface normal at the contact point (XYZ) represented by the unit vector from the contact point (XYZ)
PQR	The toolaxis represented by the unit vector from the XYZ components

Example (3-axis Tool path Output)

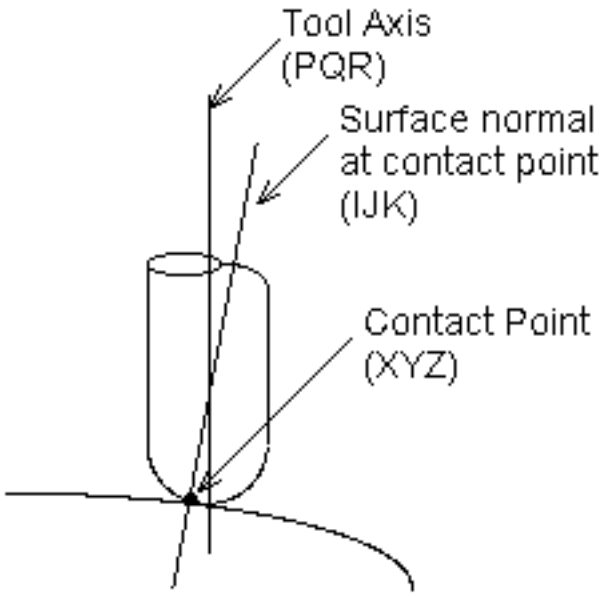
Enter the following commands:

```
NC:> MULTAX ON
NC:> DIACOMP NORMAL
NC:> SURFCUT3 :d1
NC:> DIACOMP OFF
NC:> EXIT
```

A sample output follows:

```
LOADTL 1 LENGTH 0.0000
CUTTER 1.0000 0.5000 0.0000 0.5000 0.0000 0.0000 2.0000
MULTAX ON
CUTCOM NORMDS
RAPID
GOTO   -5.2558,   -3.4778,   -3.0000,   $  (contact point XYZ)
0.4604,  -0.5551,  0.6220, $ (surface normal at contact point IJK)
0.0000,   0.0000,   1.0000     (tool axis PQR)
FEDRAT 0.0000 IPM
GOTO   -5.2558,   -3.4778,   -3.0000,   $
0.4604,   -0.5551,   0.6220,  $
0.0000,   0.0000,   1.0000
PPRINT/   START OF CUT #       1
GOTO   -7.9127,   -2.1884,   -0.0004,   $
0.6178,   -0.3051,   0.4867,  $
0.0000,   0.0000,   1.0000
PPRINT/   START OF CONNECTION
RAPID
GOTO   -7.9127,   -2.1884,   -0.0004,   $
0.6178,   -0.3051,   0.4867,  $
0.0000,   0.0000,   1.0000
etc...
```

Figure 5-11



Example (5-axis Tool path Output)

Enter the following commands:

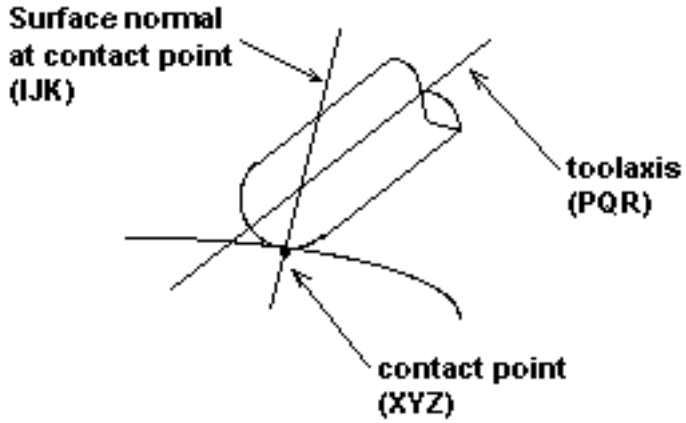
```
NC:> MULTAX ON
NC:> DIACOMP NORMAL
NC:> SURFCUT5 :d1
NC:> DIACOMP OFF
NC:> EXIT
```

A sample output follows:

```
LOADTL 1 LENGTH 0.0000
CUTTER 1.0000 0.5000 0.0000 0.5000 0.0000 0.0000 2.0000
MULTAX ON
CUTCOM NORMDS
RAPID
GOTO   -5.2558,   -3.4778,   -3.0000,   $  (contact point XYZ)
0.4604, -0.5551,  0.6220, $  (surface normal at contact point IJK)
0.5604,   -0.4551,   0.6920    (tool axis PQR)
FEDRAT 0.0000 IPM
GOTO   -5.2558,   -3.4778,   -3.0000,   $
0.4604,   -0.5551,   0.6220,  $
0.5604,   -0.4551,   0.6920
PPRINT/   START OF CUT #       1
GOTO   -7.9127,   -2.1884,   -0.0004,   $
0.6178,   -0.3051,   0.4867,  $
0.7178,   -0.1151,   0.6867
PPRINT/   START OF CONNECTION
RAPID
```

```
GOTO   -7.9127,   -2.1884,   -0.0004,   $
0.6178,   -0.3051,   0.4867,  $
0.7178,   -0.1151,   0.6867
etc...
```

Figure 5-12



Please note: Presently, you can use the SURFCUT3, SURFCUT5, and ZPROF3 commands for contact point output generation.

Calculating Tool Offset — CALCRAD

When the entity is as high as the corner radius of the tool, use CALCRAD to calculate the tool offset required to cut the entity.

When MOVE, APPROACH, PLUNGE, CUT, RETRACT, and CLEAR are used for machining and the height of an entity is the same as that of the corner radius of the tool, use CALCRAD to calculate the tool offset necessary to cut the entity.

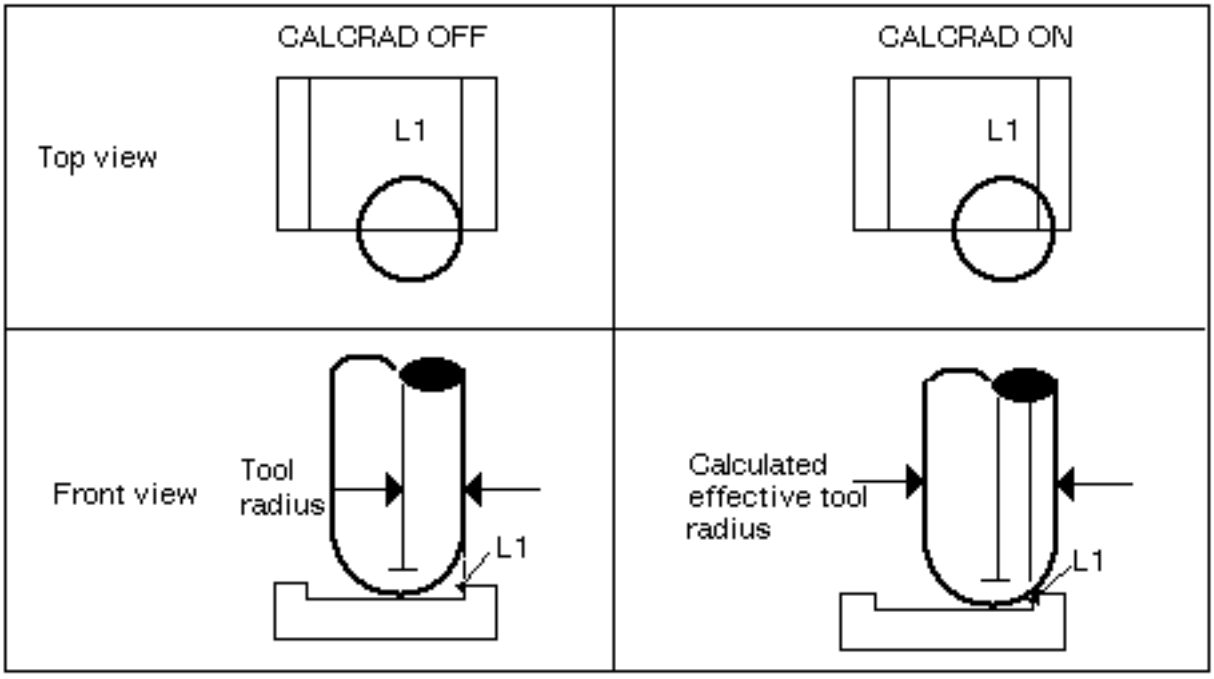
You can use either of two calculation modes:

- CALCRAD OFF (the default) positions the tool to appear tangent to the entity when viewed from above. (In reality, the tool does not touch the geometry.)
- CALCRAD ON positions the tool to actually touch the geometry. (In this case, the tool appears to violate the geometry when viewed from above.)

These conditions are shown in the example and figure given below.

```
NC:> CUT TO $L1 DIREND : Model loc d;
```

Figure 5-13 Tool Positions with CALCRAD OFF and ON

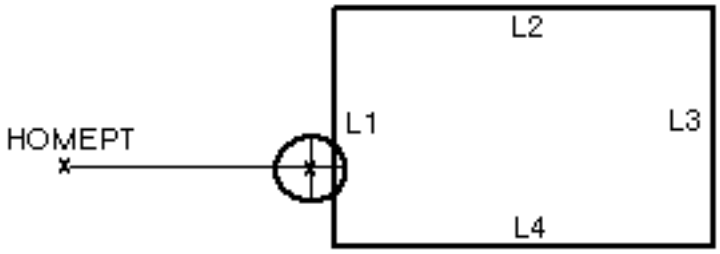


The following example shows the effect of turning CALCRAD ON and OFF between CUT ENTITY commands.

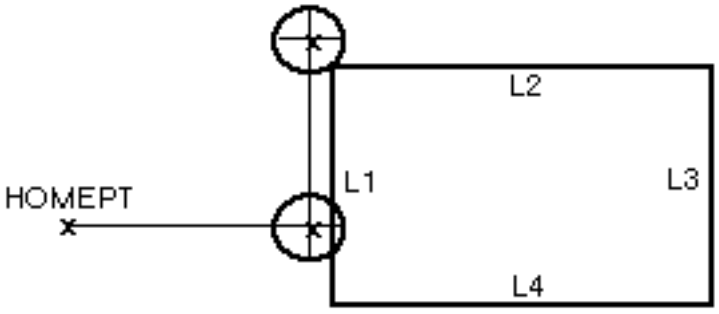
The ZWORK plane is set at -0.25 and the z-level of the entities to be cut (L1, L2, L3, and L4) are at 0. When CALCRAD is ON, the tool radius is adjusted by the difference of these z-levels.

The examples given below show how to calculate the effective tool radius and CUTs TO $L1 and CUTs PAST $L2.

```
NC:> PLANE ZWORK -0.25
NC:> CALCRAD ON
NC:> CUT TO $L1;
```

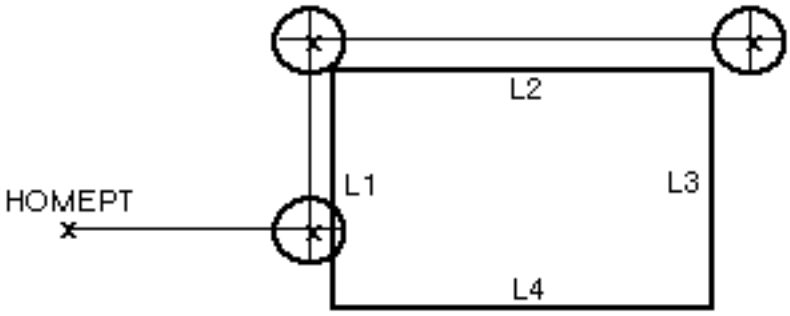



```
NC:> CUT ENTITY CHECK PAST $L2;
```



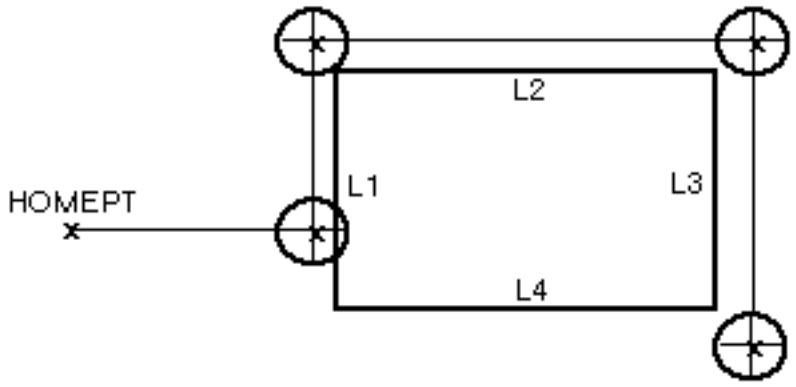
CALCRAD is now turned OFF. The next CUT is made at the same z-level. CALCRAD is still applied to L2, but not applied to the next entity L3.

```
NC:> CALCRAD OFF
NC:> CUT ENTITY CHECK PAST $L3;
```



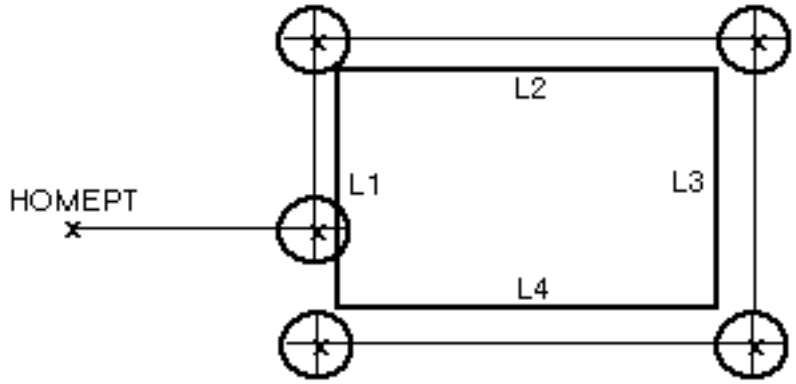
Because CALCRAD is off, the entire tool cuts PAST L4.

```
NC:> CUT ENTITY CHECK PAST $L4
```



When CALCRAD is turned back ON, it will be applied to L1, but not L4.

```
NC:> CALCRAD ON
NC:> CUT ENTITY CHECK PAST $L1;
```



Multiaxis Output — MULTAX

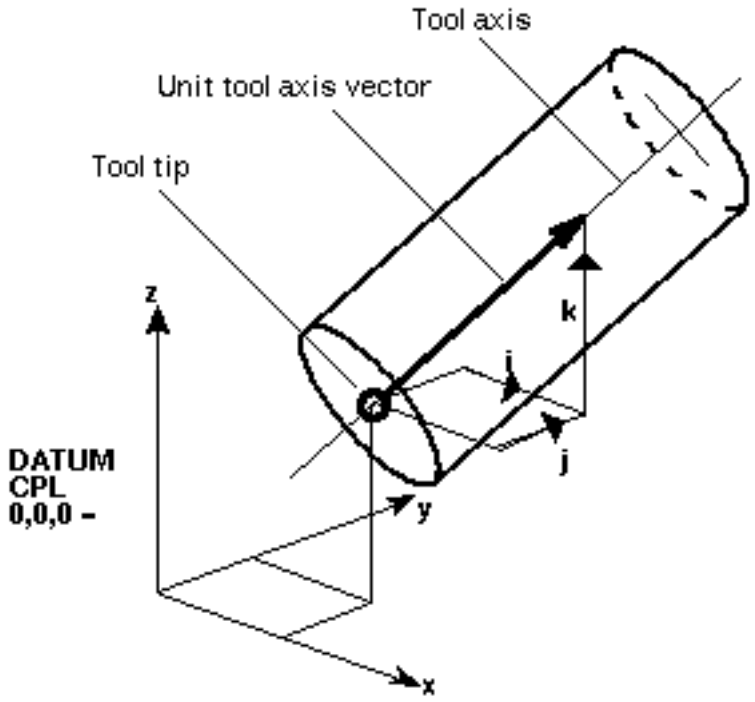
Use MULTAX to control the generation of tool axis vectors in your CLFile or APT source file. Tool axis vectors are usually unnecessary when generating 2- and 2 1/2-axis output.

Use MULTAX ON to generate tool axis vectors. Output is xyz coordinates for tool tip locations and ijk vectors for the tool axis. See the following figure. If you want to generate ROTATE and ORIGIN statements in the output, use MULTAX ON ROTATE.

Please note: In CVNC-M2, the tool tip is the center of rotation at the cutting end of the tool; this applies to all tool types.

Use MULTAX OFF (the default condition) to switch off generation of tool axis vectors. Output is xyz coordinates for tool tip locations only.

Figure 5-14 2- and 2 1/2-Axis Output Coordinates



Chapter 6

Tool Motion Generation: Milling

Use the milling commands for your positioning, cutting, profiling, pocketing, and lace cutting operations.

- Overview of Milling Commands
- Cutting with Linear or Circular Motion — CUT
- Working with the CUT Commands
- Creating Plunging Motion — PLUNGE
- Machining Around Contiguous Entities — PROFILE
- Machining Within a Closed Boundary — POCKET
- Generating Area Clearance Tool Paths — AREAMILL

Overview of Milling Commands

The milling commands support the programming of various types of positioning, cutting, profiling, pocketing, and lace cutting.

Moving the Cutter with PLUNGE and CUT

The PLUNGE and CUT commands (PLUNGE, CUT, CUT ARC, and CUT ENTITY) allow you to move the milling tool interactively through or along the part being machined.

Ordinarily, PLUNGE is used to move the tool from above the material into the material, while CUT is used to move the tool within the material once it is there.

PLUNGE causes tool motion at the PLUNGE feed rate and moves the tool to the ZWORK plane, unless you explicitly call for a different z-value.

CUT allows you to make linear motions by specifying incremental or absolute changes in x-, y-, and/or z-values relative to the current location of the tool.

CUT ARC enables you to do circular interpolation along a described arc. CVNC-M2 generates circular interpolation based on your specification of the center point, radius, start and end angles, and cutting direction.

CUT ENTITY lets you move the tool along one or more entities. If more than one drive entity is selected, motion is generated sequentially along one entity until the next entity is encountered.

A series of CUT ENTITY commands can be combined with changes in stock offset (see Chapter 5 on STOCK), cutter compensation, or other commands used to specify operational parameters or motion. CUT ENTITY can machine lines, arcs, conics, B-splines, Nsplines, and Cpoles.

Rough Cutting and Finishing with PROFILE and POCKET

The PROFILE and POCKET commands provide automatic operations designed to rough and finish a part with minimum user interaction.

A pocket or profile boundary is defined by a set of entities that can be grouped together as an NCGROUP or specified individually. A profile boundary may be open or closed; a pocket boundary must be closed.

PROFILE and its macro variations (DPROF and SPROF) allow you to machine around a contiguous string of part entities, treating the entire collection as a single boundary.

PROFILE does not allow changes in stock offset, cutter compensation, or ON/TO modes (as does CUT ENTITY). By treating the string of entities as a single boundary, it avoids entry into narrow slots or corners where the tool does not fit.

Machining Profiles and Pockets with Macros

The DPROF macro machines several profiles at an incremental depth value on or tangent to a profile boundary. It also lets you machine several profiles at an incremental depth and angle value (which determine material offsets) on or tangent to a profile boundary.

The SPROF macro lets you machine several profiles at an incremental stock value tangent to a profile boundary.

POCKET and its macro variation, DPOCK, machine the entire area within a closed boundary (comprising a string of part entities). The tool maintains a constant cutter/material relationship while milling the area to be machined. You may identify and isolate up to 20 islands comprising part entities within a pocket that the tool will avoid during machining.

The DPOCK macro machines a pocket with multiple passes at different depths. If you want to machine an angular pocket, DPOCK will calculate increasing material offsets at each step-down tool pass.

Area Clearance

AREAMILL generates lace cut and contour tool paths to clear an area within a boundary and around islands.

Cutting with Linear or Circular Motion — CUT

CUT, CUT ARC, and CUT ENTITY move the cutting tool to locations you specify.

The CUT command cuts material; use it to move the cutting tool from its current location to a new location.

The default CUT feed rate is 0. You must set a feed rate with the FEED command before using CUT. CUT moves at the CUT feed rate at the current z-value unless you specify a different z-value.

The default coordinates of the new cut location are the same as those of the previous location unless you specify otherwise. For example, if your current location is x4, y4, z4 and you enter `CUT XABS 3`, the resulting location will be x3, y4, z4.

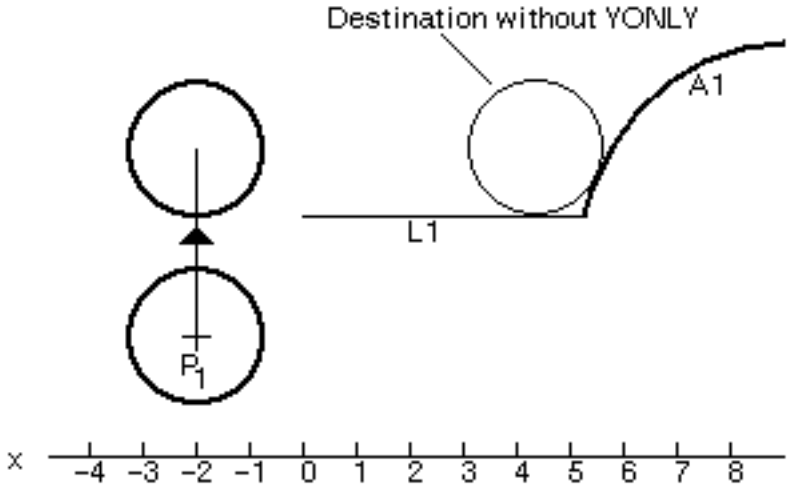
CUT makes linear cutting motions by specifying incremental or absolute changes in x-, y-, and/or z-values relative to the current location of the tool. (CUT ARC cuts in a circular motion.)

CUT motion may also be to, on, or past a specified entity.

For example, to start at the P1 location and cut in the y-direction past $L1 to the intersection of $A1, enter

```
NC:> CUT YONLY PAST $L1; INTOF TO $A1
```

Figure 6-1 Tool Motion in the y-Direction Using the YONLY Modifier



The following examples show the effects on tool motion of different CUT modifiers. (The same modifiers and effects apply to the APPROACH, MOVE, PLUNGE, CLEAR, and RETRACT commands.) In each example, the current tool position is designated by P_1.

Figure 6-2 Tool Motion in the x-Direction Using XONLY

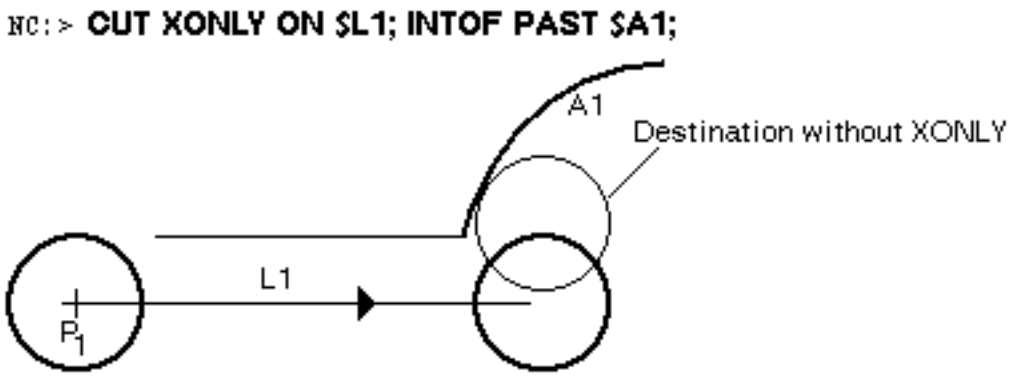


Figure 6-3 Tool Motion in the y-Direction Using YONLY

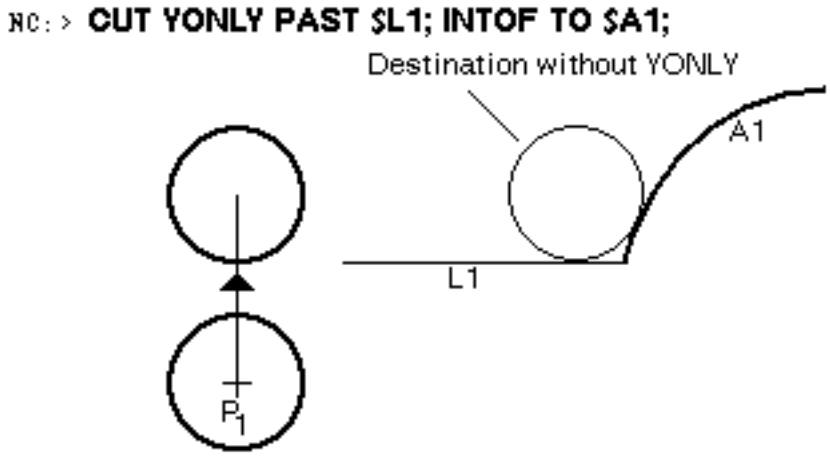


Figure 6-4 Tool Motion with XABS

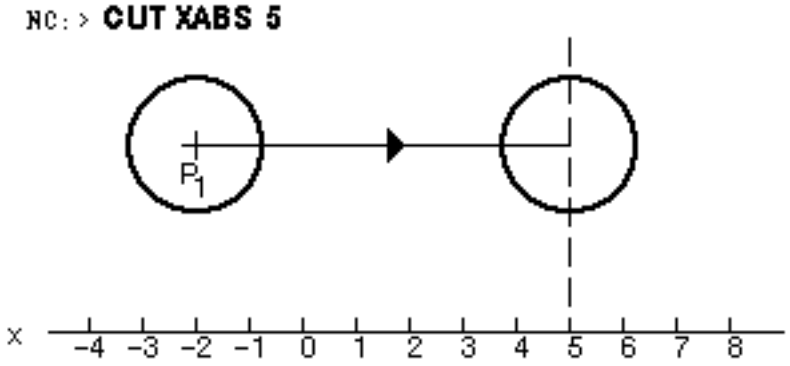


Figure 6-5 Tool Motion Using XINC

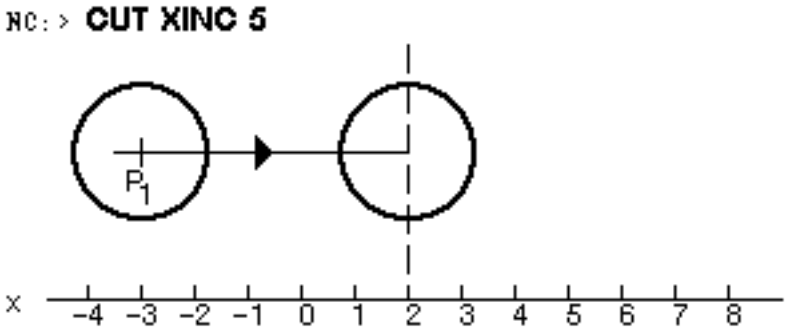


Figure 6-6 Tool Motion Using DIRLIN

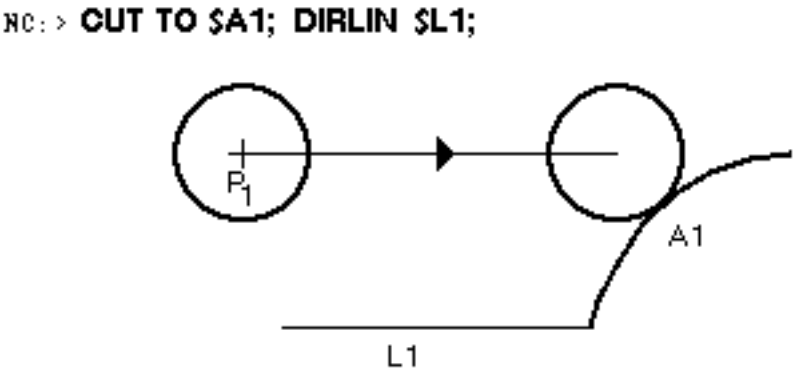


Figure 6-7 Tool Motion Using DIRLOC

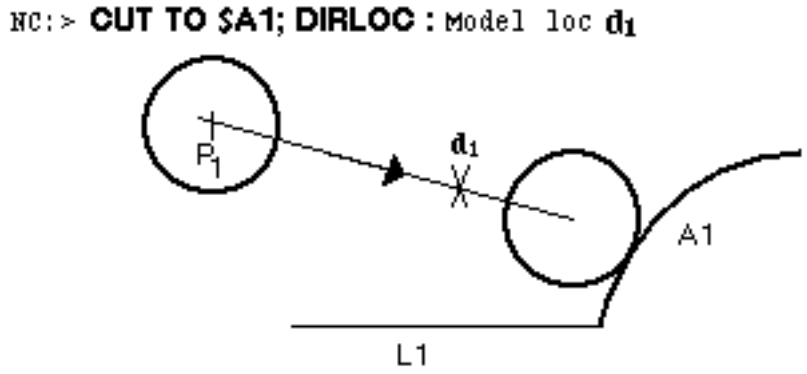


Figure 6-8 Tool Motion Using DIREND

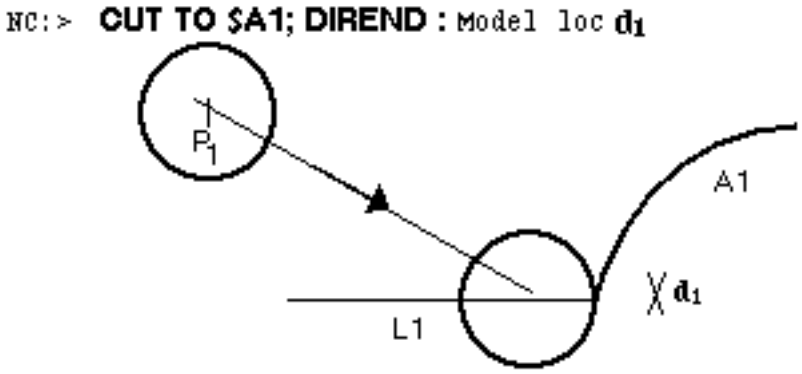


Figure 6-9 Tool Motion Using DIRVEC

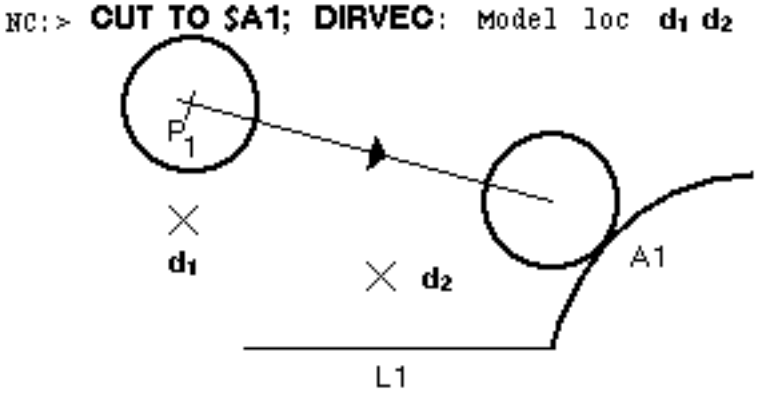


Figure 6-10 Tool Motion Using NORMAL

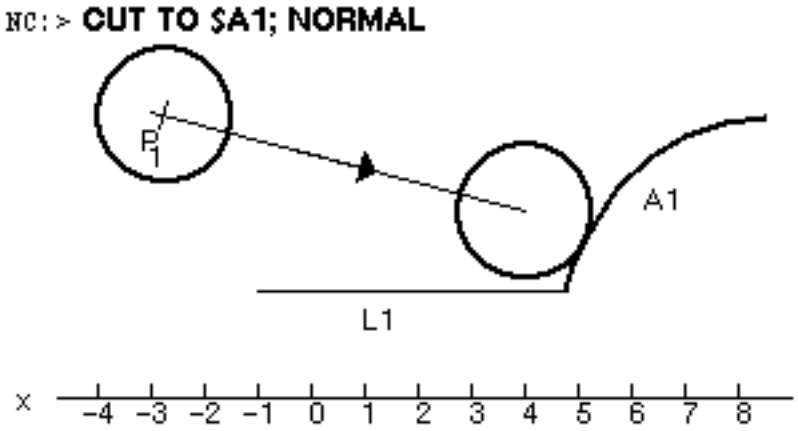
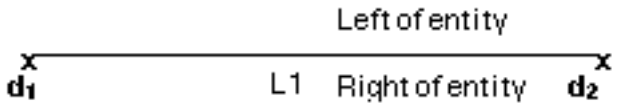


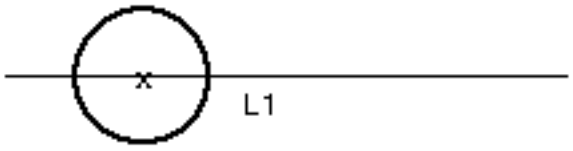
Figure 6-11 Tool Motion to Left of Entity Using CUT TO

OnINSert LINe: MODEL loc **d_1 d_2;**

1. L1 created from left to right

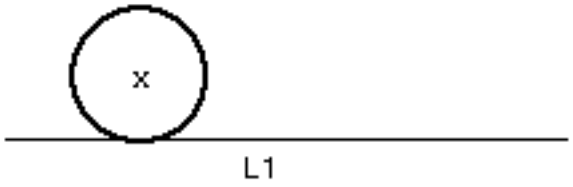


2. Tool in ON position



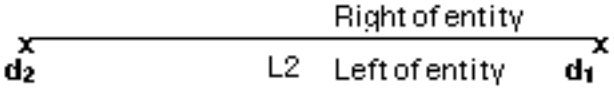
NC:> **CUT TO $L1**

3. CUTs to left of L1

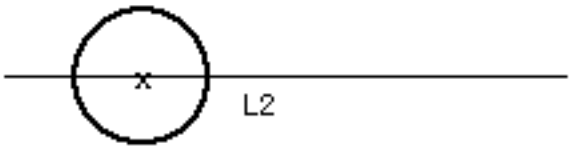


OnINSert LINe: MODEL loc **d_1 d_2;**

1. L2 created from right to left



2. Tool in ON position



NC:> **CUT TO $L2**

3. CUTs to left of L2

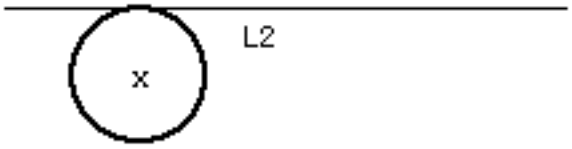
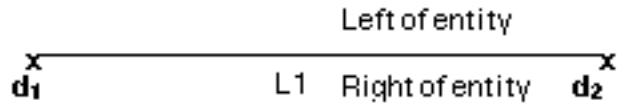


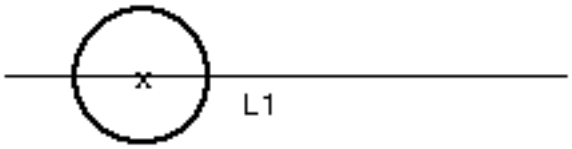
Figure 6-12 Tool Motion to Right of Entity Using CUT PAST

OnINSert LINe: MODEL loc **d_1 d_2;**

1. L1 created from left to right

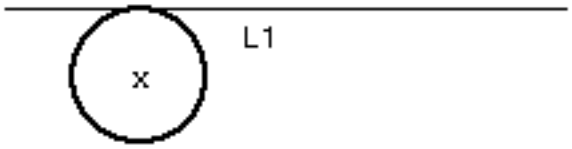


2. Tool in ON position



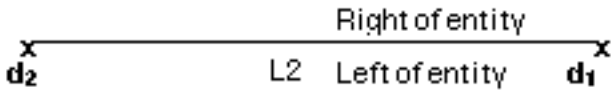
NC:> **CUT PAST $L1**

3. CUTs to right of L1

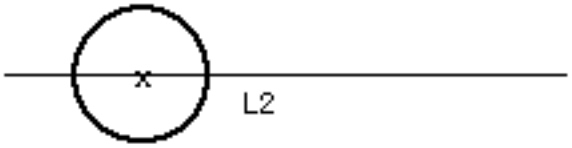


OnINSert LINe: MODEL loc **d_1 d_2;**

1. L2 created from right to left



2. Tool in ON position



NC:> **CUT PAST $L2**

3. CUTs to right of L2

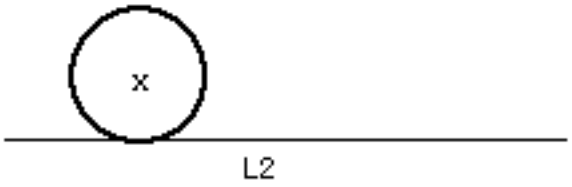


Figure 6-13 Tool Motion Using INTOF TO

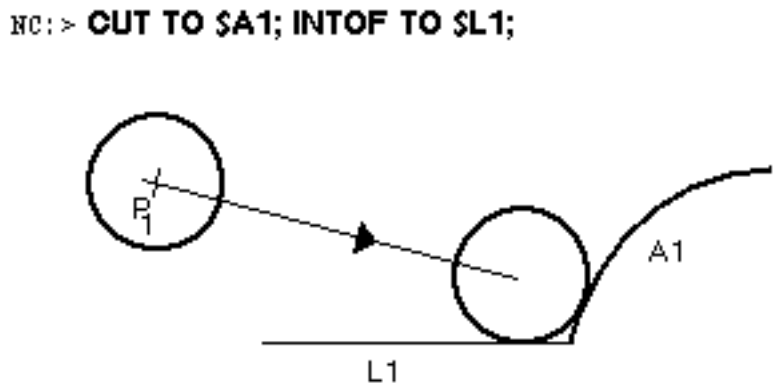
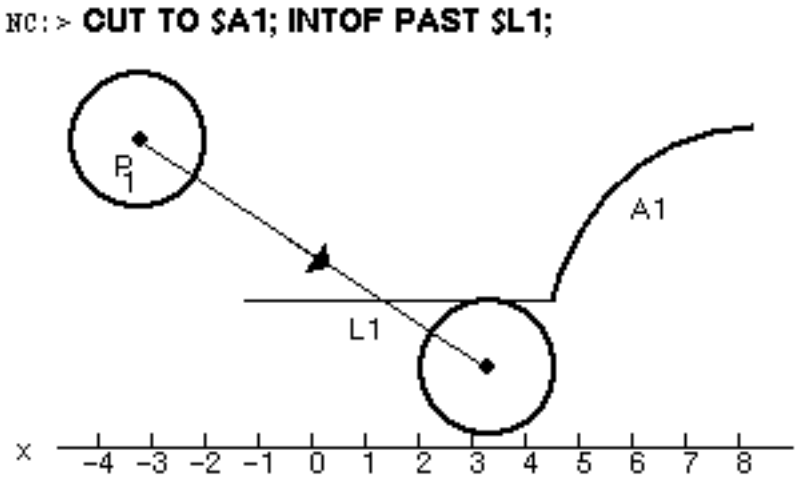


Figure 6-14 Tool Motion Using INTOF PAST



Generating Circular Interpolation — CUT ARC

Use the CUT ARC command to move the cutter along an arc without referencing CADDS geometry.

CUT ARC performs circular interpolation along a described arc.

CVNC-M2 generates circular interpolation based on your specification of the center point, radius, start and end angles, and cutting direction.

Please note: If the start and end angles are the same, CUT ARC generates a full circle.

The tool tip moves at the CUT feed rate (set with FEED) to the first point of the arc, then proceeds on the arc from the start angle to the end angle. For example,

```
NC:> CUT ARC d1 RADIUS .005 AGO 45 AEND 90 CLW
```

Moving the Tool Along an Entity — CUT ENTITY

CUT ENTITY moves the cutting tool along a specified drive entity from its current location to, on, or past a specified entity.

CUT ENTITY moves the tool along one or more entities. If you select more than one drive entity, CUT ENTITY generates motion sequentially along one entity until the next is encountered. You must specify the drive entity using motion TO, ON, PAST, or INTOF.

Use STOP to continue using the current drive entity; use CHECK to change the drive entity to the entity with the TO, ON, or PAST condition.

Please note: CHECK is required for the first pass and optional thereafter.

A series of CUT ENTITY commands can be combined with changes in stock offset (see Chapter 5 on STOCK), cutter compensation, or other commands used to specify operational parameters or motion. CUT ENTITY can machine lines, arcs, conics, B-splines, Nsplines, and Cpoles.

You can use a bias location with TO or PAST to position the tool tangent to the selected entity. If you use TO, the tool positions on the same side of the entity as the bias point. If you use PAST, the tool positions on the side opposite the bias point.

When machining an arc, the bias point

- Determines the TO/PAST condition, as described earlier.
- Indicates at which intersection the tool will check.

If not specified, the current tool location serves as a BIAS point.

If the specified drive entity is an arc or a circle, you must specify the direction of motion as CLW or CCLW.

The following examples show how to use CUT ENTITY with the CHECK TO, CHECK ON, and CHECK PAST modifiers.

Figure 6-15 Tool Motion Using CUT ENTITY CHECK TO

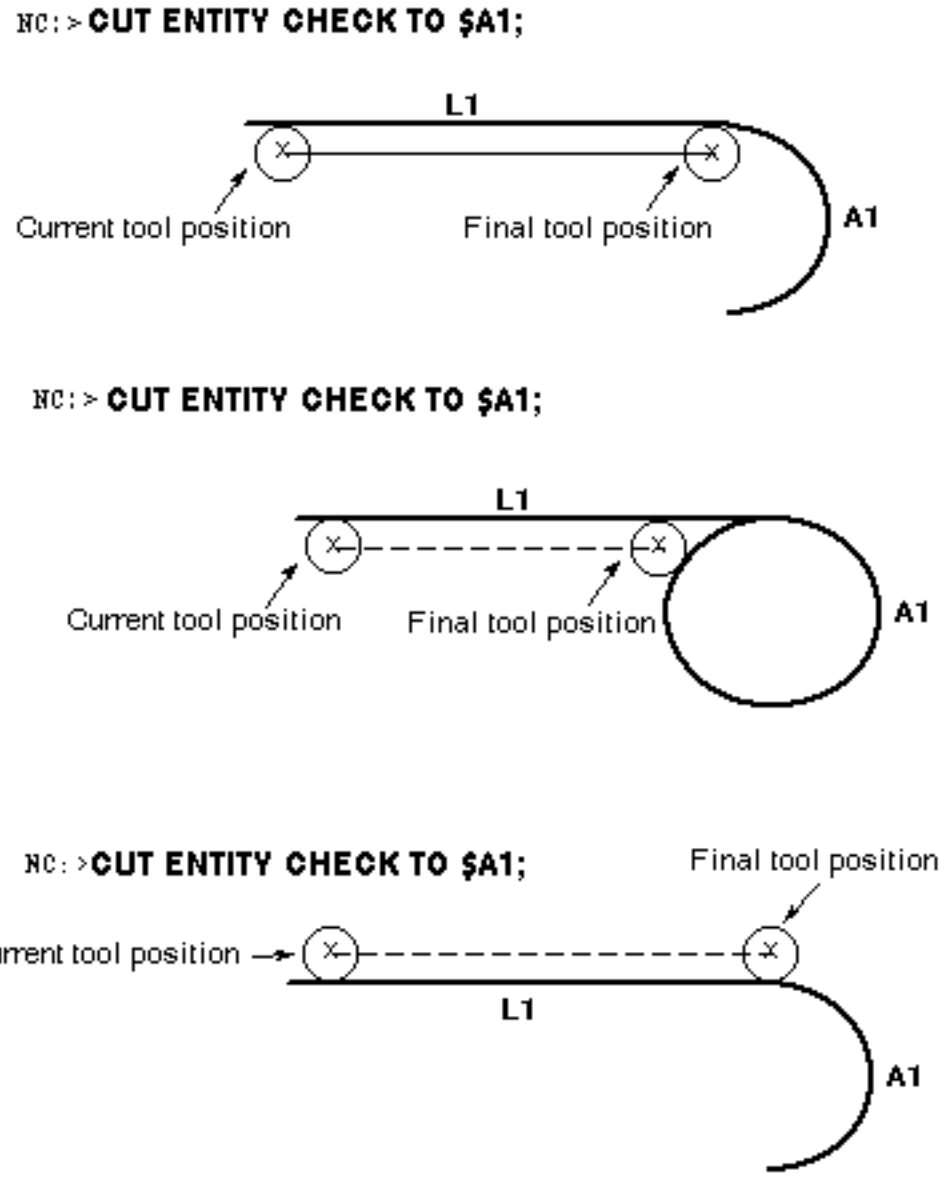
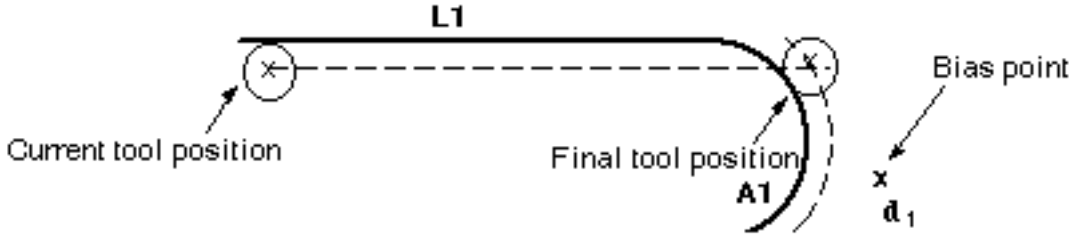
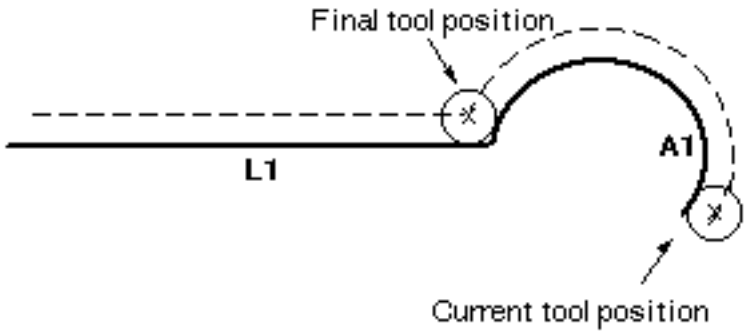


Figure 6-16 Tool Motion Using CUT ENTITY CHECK TO (continued)

NC:>**CUT ENTITY CHECK TO $A1; BIAS** Model loc d_1



NC:>**CUT ENTITY CHECK TO $L1;**



NC:>**CUT ENTITY CHECK TO $A1; BIAS** Model loc d_1

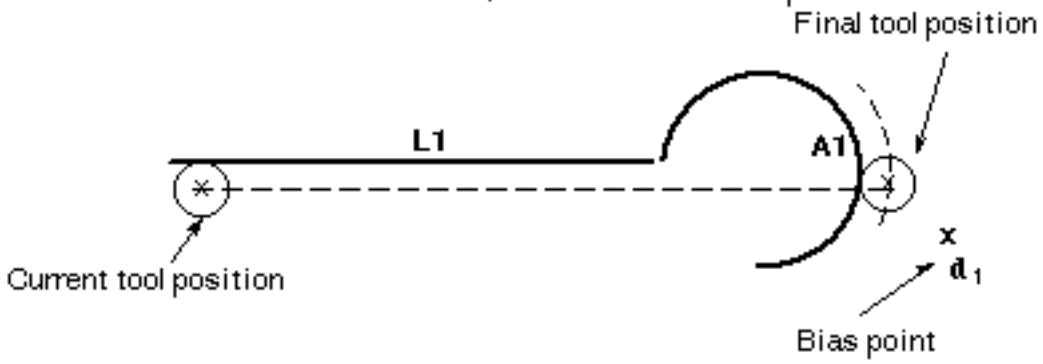


Figure 6-17 Tool Motion Using CUT ENTITY CHECK ON

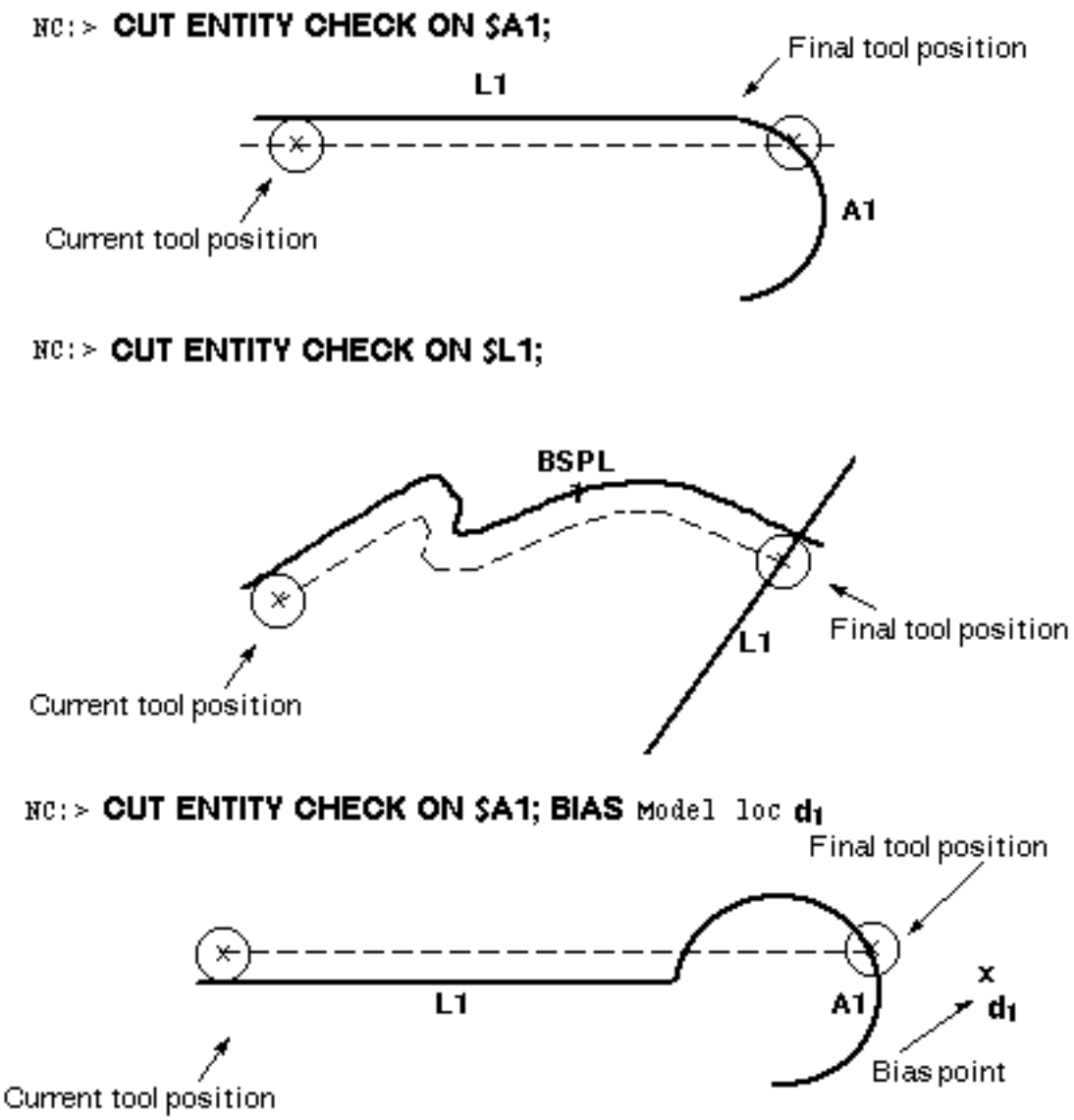
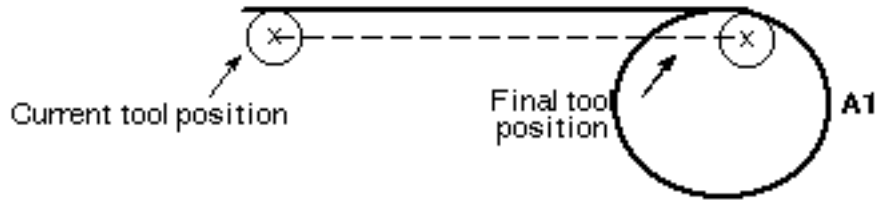
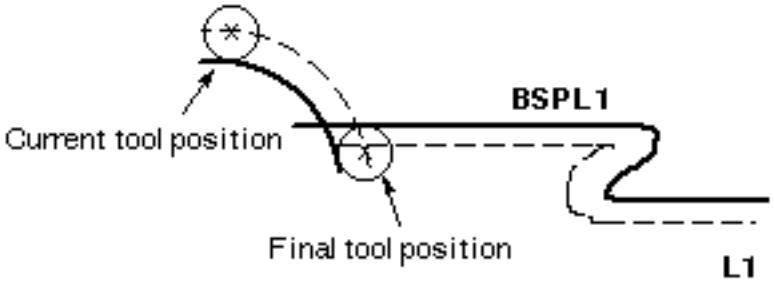


Figure 6-18 Tool Motion Using CUT ENTITY CHECK PAST

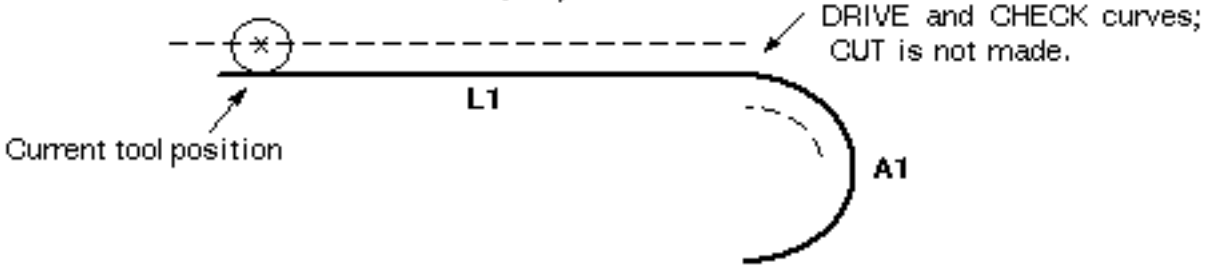
NC:>**CUT ENTITY CHECK PAST $A1;**



NC:**CUT ENTITY CHECK PAST $BSPL1;**

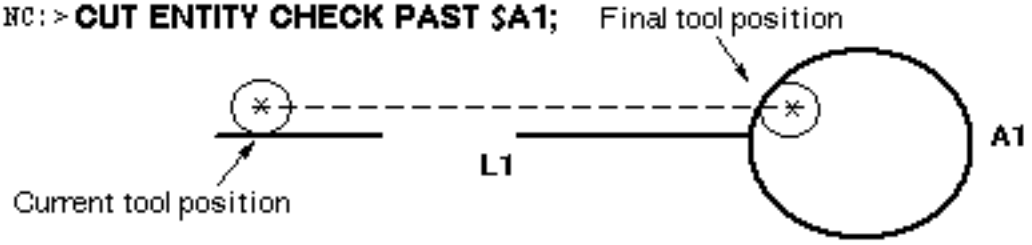


NC:> **CUT ENTITY CHECK PAST $A1;**



Note that CUT ENTITY CHECK TO $A1 *would* work in this case.

NC:> **CUT ENTITY CHECK PAST $A1;**



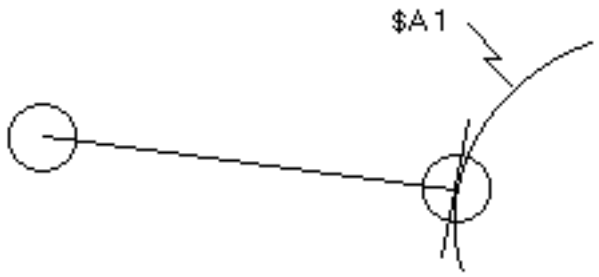
Working with the CUT Commands

Your CUT command modifiers will differ, depending on the points of normalcy, points of intersection, and the side of the boundary to be machined.

CUT ON - Single Point of Normalcy

CUT ON positions the tool center to the “nearest normal” point on the entity being cut. From this point, a normal line can be drawn that intersects the current location of the tool. This is illustrated as follows.

```
NC:> CUT ON $A1 ;
```



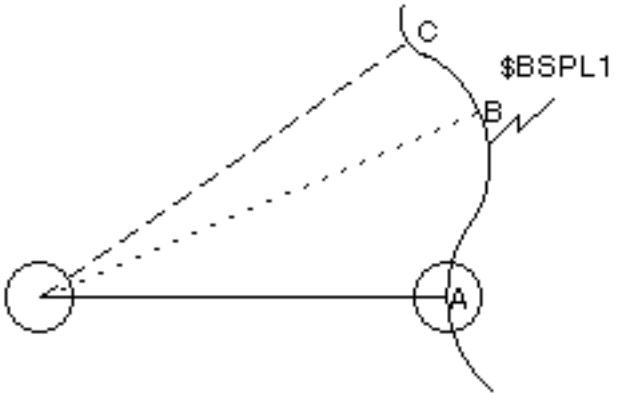
CUT ON - Multiple Points of Normalcy

A nearest normal point is a normal point where, in either direction from that point, the distance to the current tool location would be greater (points A and C in the next illustration).

In the event of more than one nearest normal point on the entity to be cut, the cut is performed to the closest “nearest normal” point (point A).

There may also be normal points not considered nearest normal in which, in either direction from that point, the distance to the current tool location would be less (point B). These “farthest normal” points are never positioned to, even if the only normal on a curve is the farthest normal. See the next section, “CUT ON - No Points of Normalcy” on page 6-18.

```
NC:> CUT ON $BSPL1 ;
```

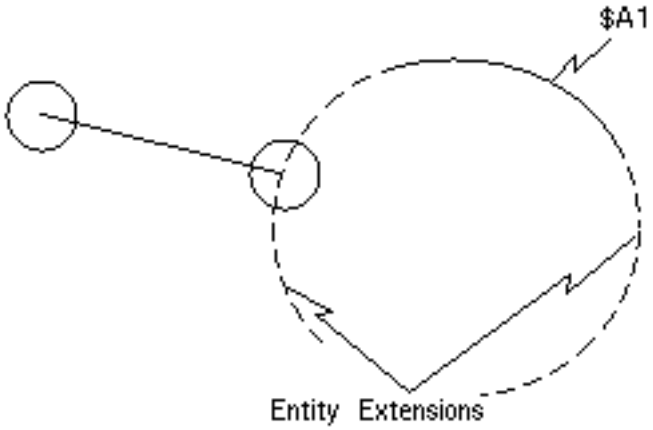


CUT ON - No Points of Normalcy

If there are no nearest normal points, the entity is extended until a nearest normal point is found. The extensions do not appear on the screen.

In the following example, the extensions created two points of normalcy at 9:30 and 3:30 (hour-hand position on a clock at 9:30/3:30), but 9:30 was selected because it is a nearest normal point.

```
NC:> CUT ON $A1 ;
```



Please note: There are two conditions under which extensions are applied:

- A nearest normal point does not exist on the current entity. In this case, the entity is extended to create a nearest normal point.

- An intersection is required between two entities that do not intersect. Extensions are applied to both entities simultaneously to create an intersection point.

All entities are treated as parametric curves. Extensions are created by parametrically extending a curve 25% on each end. After this extension, CVNC checks to see if the condition being searched for is satisfied (point of normalcy or intersection).

If the condition is still not satisfied, the curve is again extended by 25% on each end. The curve can be extended in this way up to three times. A line will be extended to 100 times its length (after the previous three attempts have failed) in an attempt to satisfy the condition.

If a point of normalcy is not found, the closest point on the extended curve is used. If an intersection point is not found, an error is returned.

CUT TO/CUT PAST - Determining Side-of-Boundary

When cutting TO or PAST, the tool is placed tangent to the entity at the point of normalcy (as determined by using the previous example's methods).

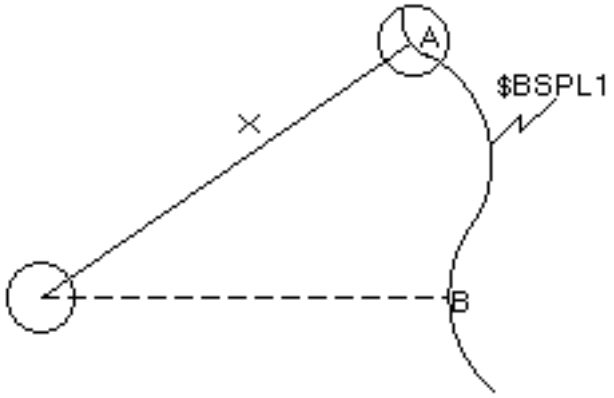
The TO side of the entity is the side of the entity at the point of normalcy nearest to the current tool location.

Bias Points

Bias points are used at two different junctures when processing the CUT command:

- When determining the point of normalcy (with multiple points of normalcy), the normal point (A) that is nearest the bias point is used instead of the one (B) nearest the current tool location. See "CUT ON - Multiple Points of Normalcy" on page 6-17.

```
NC:> CUT ON $BSPL1 ;BIASPNT X 3 Y 2 ;
```



- The bias point is used instead of the current tool location when determining the side-of-boundary. This is shown as follows.

```
NC:> CUT TO $L1 ;BIASPNT X 6 Y 0 ;
```

CUT ENTITY

CUT ENTITY STOP is equivalent to CUT STOP. CUT ENTITY CHECK is equivalent to CUT CHECK.

CUT CHECK ON - One Intersection Point

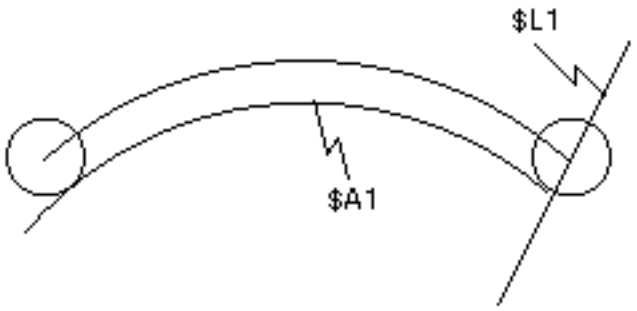
Please note: You must precede CUT CHECK with a CUT or MOVE command designating an entity as the DRIVE CURVE.

CUT CHECK ON moves the tool on or along the drive entity (and possibly its extension) until the center point of the tool is positioned on the check entity.

Whether the tool moves on or along the drive entity depends on whether the previous CUT command positioned the tool on or to/past the drive curve.

```
NC:> CUT TO $A1 ;
```

```
NC:> CUT CHECK ON $L1 ;CLW
```

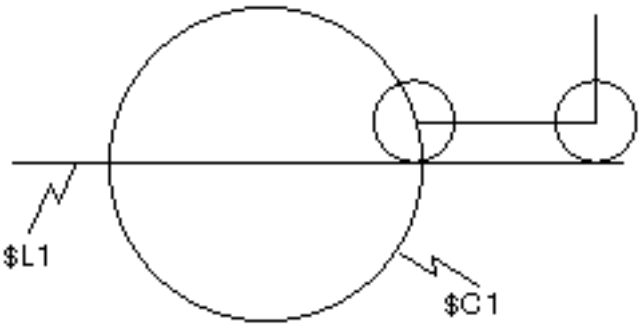


CUT CHECK ON - Multiple Intersection Points

When multiple intersection points occur between the drive and check curve, the best intersection is used. The best intersection is defined as the intersection point closest to the current tool location.

If the BIASPNT modifier is used, the best intersection is the intersection point closest to the bias point.

```
NC:> CUT TO $L1 ;
NC:> CUT CHECK ON $C1 ;
```



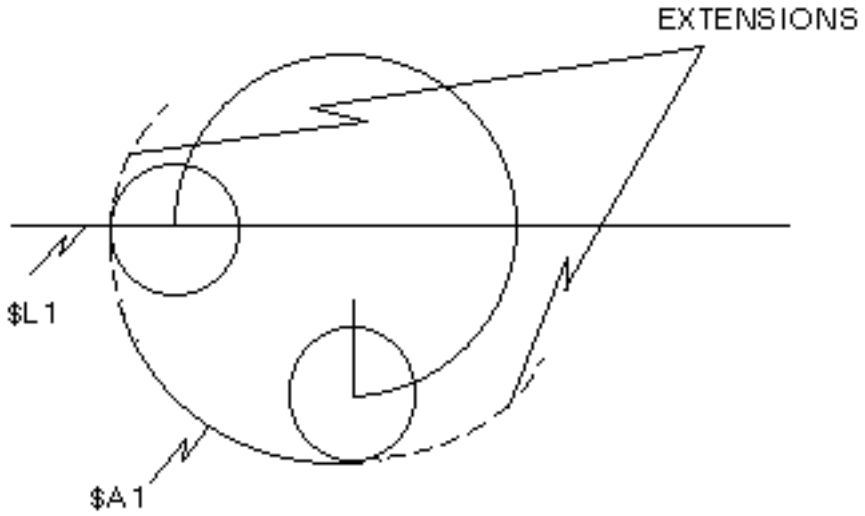
CUT CHECK ON - No Intersection Points

When no intersection points occur between the drive and check curve, both curves are extended until one or more intersections are found. If more than one intersection is found, CVNC determines the best intersection.

If no intersection is created by extending the curves, an error message is displayed. See the previous section, “CUT ON - No Points of Normalcy” on page 6-18.

The following example shows how you might get a result you do not expect. Because of the incremental extensions of the arc (which are not treated as circles), the 3:00 intersection is not found, and a cut is made to the 9:00 intersection. To assure the proper result, use intersecting geometry when performing CUT CHECKs.

```
NC:> CUT TO $A1 ;
NC:> CUT CHECK ON $L1 ; CCLW
```



CUT CHECK TO/PAST - Determining Side-of-Boundary

The TO side of the check entity is the side at the point of normalcy nearest to the current tool location. When the BIASPNT modifier is used, the TO side of the check entity is the side at the point of normalcy nearest the bias point.

A special algorithm allows you to enter CUT CHECK TO commands that drive along the drive entity until the tool meets the check curve. (No gouging actually occurs.) This method is not applied if a bias point is used or if the check entity is a line.

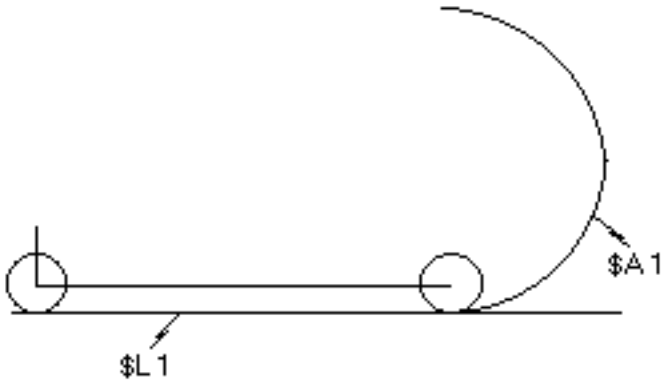
Please note: The CUT command uses bounded geometry. With CADDS 5, an arc is treated as an arc (in fact, a NURB).

CUT CHECK

In the next example, the arc is treated as a simple NURB and the tool positions to the desired side of the arc.

Note that no nearest normal point is found on the check curve.

```
NC:> CUT TO $L1 ;
NC:> CUT CHECK TO $A1 ;
```



Please note: The use of a normal point routine (given a point and a NURB) returns all nearest normal points on that NURB.

If you get unexpected results, CVNC may be unable to find the farthest normals. CUT may then position to extended portions of a curve (instead of to a desired normal position), or position to a TO/PAST side-of-boundary other than that desired. To avoid this situation, use the BIAS POINT modifier in any cases of potential ambiguity.

Creating Plunging Motion — PLUNGE

Use PLUNGE to move your milling tool along the part you are machining.

Use the PLUNGE command to cut material. PLUNGE moves the milling tool down to the z-work plane along the part being machined; it moves the tool from its current location to the new location at the PLUNGE feed rate.

If you do not specify a PLUNGE feed rate with FEED, PLUNGE defaults to the value set for the CUT feed rate. See the information on the CUT command in this chapter for examples of TO/ON/PAST and INTOF motion.

With its choice of modifiers, PLUNGE enables you to describe the location of your next cut in absolute or incremental terms, or using getdata.

You can specify the following:

- That the motion continue to, on, or past a selected entity
- The direction of the motion

For example, to specify a location in absolute values with the direction of motion normal, enter

```
NC:> PLUNGE XABS 5.0 YABS 4.4 ZABS 0.0 TO NORMAL
```

Machining Around Contiguous Entities — PROFILE

PROFILE performs boundary machining on a stated number of entities.

PROFILE provides automatic operations designed to rough and finish a part with minimum user interaction. You can use PROFILE to perform boundary machining on a specified number of entities (up to 512) in the order of their selection.

The contour is offset by the STOCK DRIVE (see Chapter 5) value in the direction of the current position or the XYSIDE selection. Indicate the xy side of the boundary by selecting a point as close as possible to the beginning of the first entity chosen.

A profile boundary is defined by a set of entities that can be specified individually or grouped as an NCGROUP [SYS].

In an open boundary, the start point in the first entity is different from the final point in the last entity. In a closed boundary these points are the same. A profile boundary may be open or closed.

Please note: If a boundary contains a series of lines that approximate an arc within machining tolerance (INTOL + OUTTOL, as defaulted or defined with TOLER), PROFILE outputs arc motion.

PROFILE and its macro variations (DPROF and SPROF) machine around a contiguous string of part entities, treating the entire collection as a single boundary.

PROFILE uses assigned stock values only if the TO modifier is chosen. PROFILE does not allow changes in stock offset, tool compensation, or ON/TO modes (as does CUT ENTITY). By treating the string of entities as a single boundary, it avoids entry into narrow slots or corners where the tool does not fit.

You can also use the FACE modifier to select a face or faces of a solid. For example,

```
NC:>PROFILE ON FACE :edges or surfs d;
NC:>PROFILE TO FACE :edges or surfs d;
```

CLW or CCLW with FACE enables you to select the face entities in the clockwise or counter-clockwise directions, respectively. In addition you can use the STARTPT and ENDPT modifiers to trim the tool path at the beginning and/or end, respectively. STARTPT is mutually exclusive with STARTLIN and ENDPT is

mutually exclusive with CHECKLIN. For more information on STARTLIN and CHECKLIN, see the *CVNC Milling Command Reference*.

Please note: CLW and CCLW can be used only with the FACE modifier and are mutually exclusive. When you use the FACE modifier, CVNC decides the LEFT and RIGHT sides according to your choice of CLW or CCLW (see Figure 6-22, “FACE with CLW and CCLW,” on page 6-29 for details).

STARTPT/STARTLIN and ENDPT/CHECKLIN can be used with or without FACE, as explained below.

- With FACE

  In this case STARTPT defines the starting entity and the direction of the tool path is based on your choice of CLW or CCLW.

- Without FACE

  In this case STARTPT and ENDPT define the direction of the tool path, if you have specified both. However, if you specify only one of them, the direction is based on the order in which you have selected the entities and starts or ends at the STARTPT or ENDPT, respectively. Use the REV modifier if you select the STARTPT and ENDPT (and thus the machining direction) in an order opposite to the order in which you have selected the entities.

Please note: If STARTLIN, CHECKLIN, STARTPT or ENDPT are not used, the cutter machines the entire length of the input boundaries and you do not have control over the start and end points (for the FACE option).

The following examples show how to use the PROFILE command.

Figure 6-19 Open and Closed Boundaries, with and without CLOSED Modifier

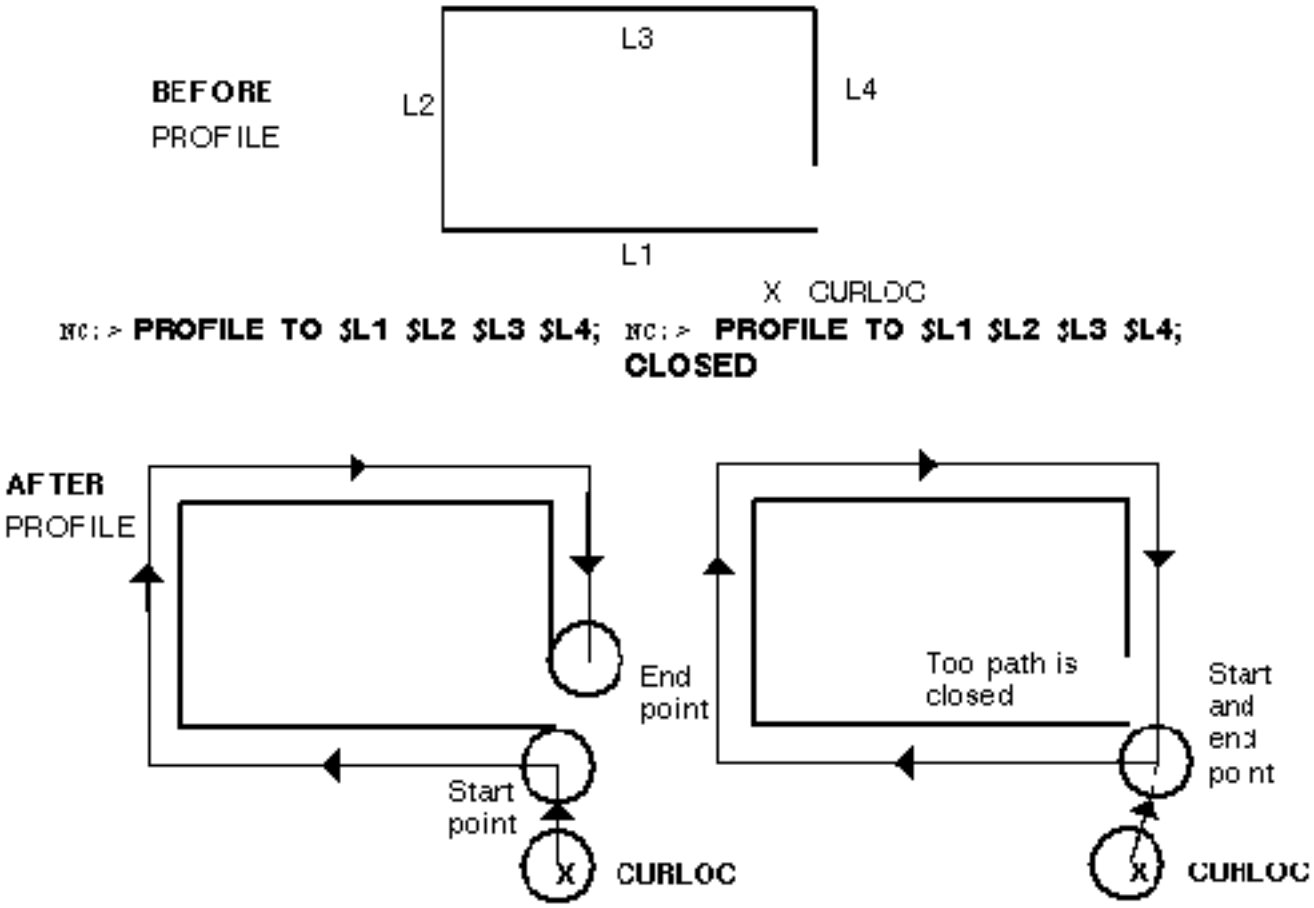


Figure 6-20 Closed Boundary, with and without CLOSED Modifier

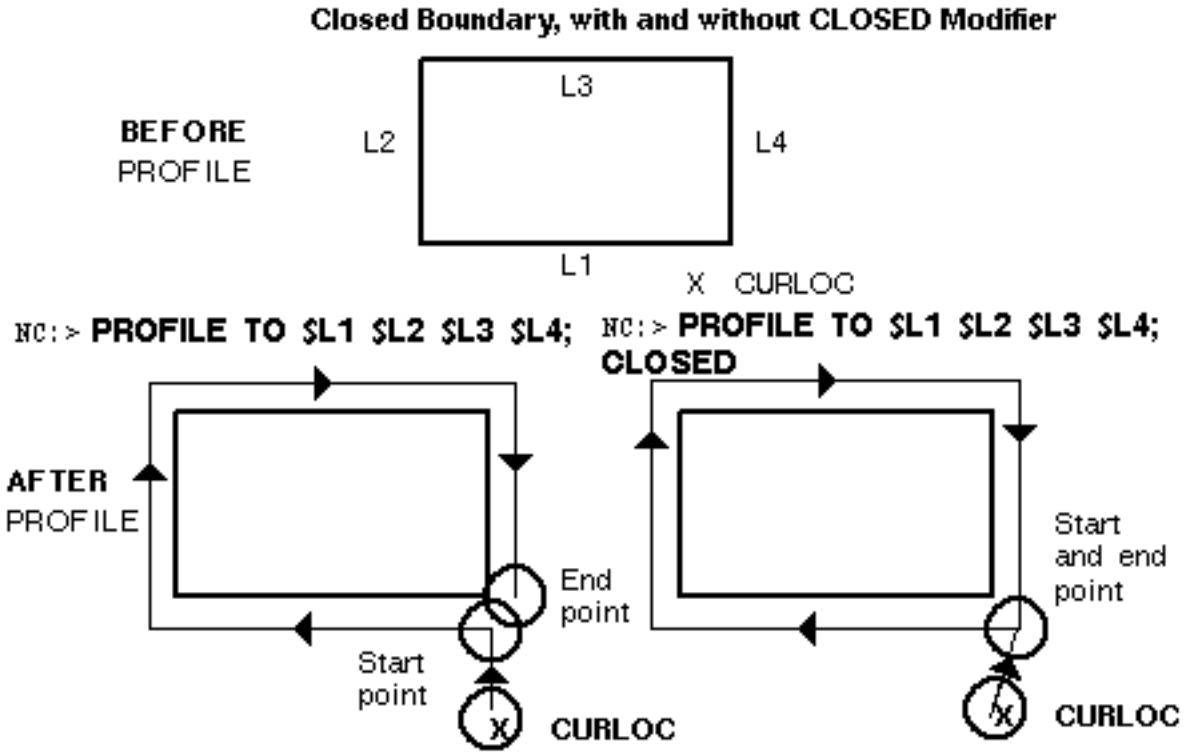


Figure 6-21 Closed Boundary with STARTPT and ENDPT

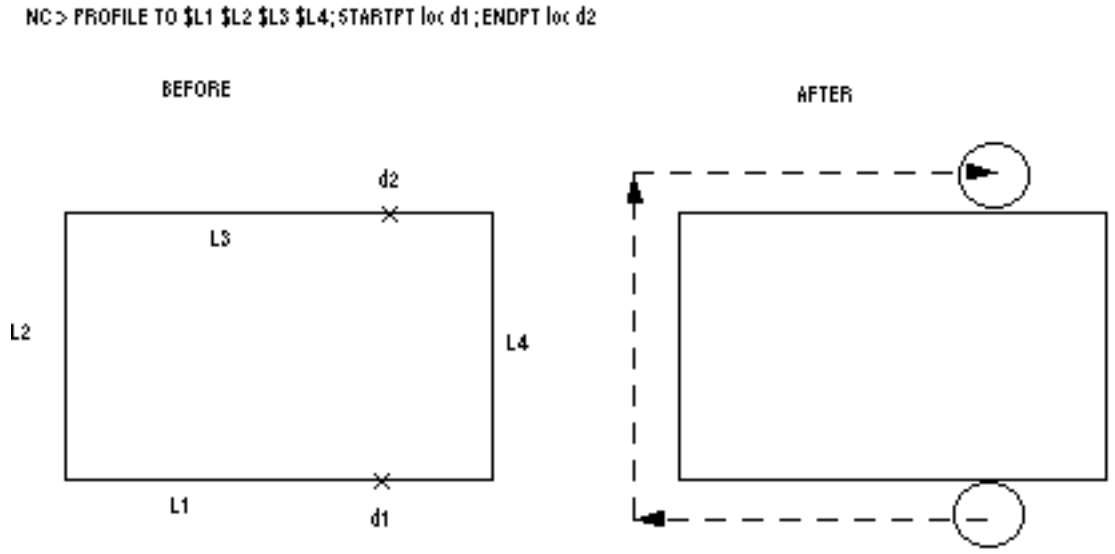
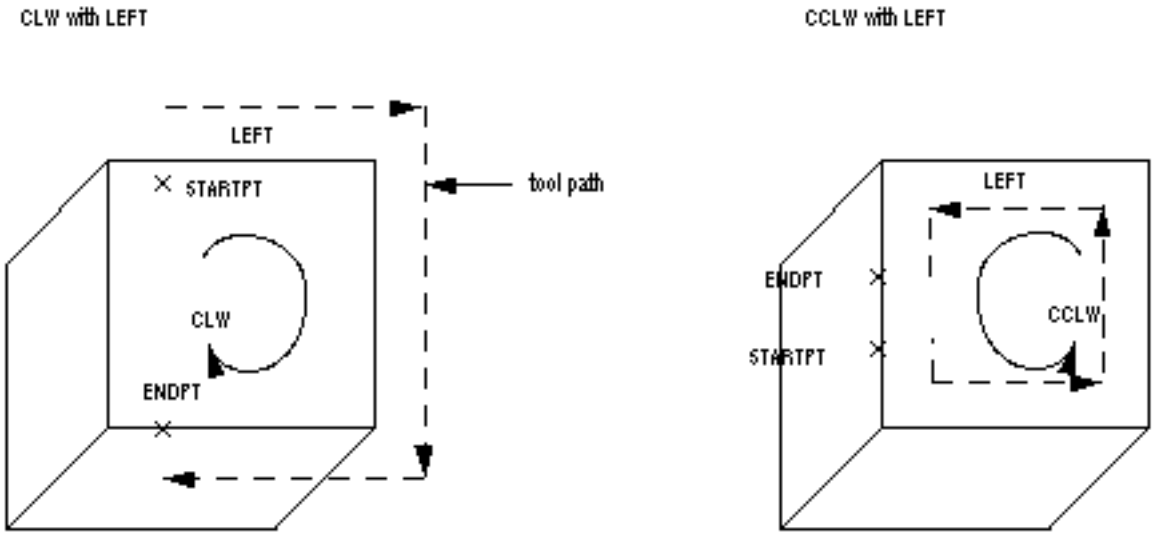


Figure 6-22 FACE with CLW and CCLW



Please note: In the above figure, the z-plane and the entities should be on the same level.

Machining Several Profiles: Depth Value — DPROF Macro

The DPROF macro generates multiple profile paths to an entity boundary at incremental depth values and at a specified angle.

The DPROF macro machines several profiles at an incremental depth value on or tangent to a profile boundary.

DPROF also machines several profiles at an incremental depth and angle value (which determine material offsets) on or tangent to a profile boundary.

DPROF allows for a tool corner radius offset.

For example, to profile an NCGROUP, enter

```
NC:> DPROF NCGROUPname CLOSED NOROLL
```

In this example, the initial contact point on each profile boundary path will be the same as the last point on that path. In addition, radial motion will be prevented at the profile corners. For more information about macros, see *Customizing CVNC*.

Machining Several Profiles: Stock Value — SPROF Macro

SPROF generates multiple profile paths at incremental stock values, until and including the final stock.

The SPROF macro allows you to machine several profiles at an incremental stock value tangent to a profile boundary.

For example, to machine profile an NCGROUP, enter

```
NC:> SPROF NCGROUPname OPEN ROLL
```

In this example, the initial contact point on the first entity of the profile boundary will be different from the last point on the last entity of the profile boundary. In addition, radial motion will be enabled at the profile corners. For more information about macros, see *Customizing CVNC*.

Machining Within a Closed Boundary — POCKET

Use the POCKET command to clear an area bounded by up to 512 entities.

POCKET clears an area bounded by a set of specified entities, machining those entities in the order of their selection. POCKET machines entities at the ZWORK plane. It allows up to 20 islands inside the pocket boundary.

A pocket boundary is defined by a set of entities that can be grouped as an NCGROUP [SYS] or specified individually.

In an open boundary the start point in the first entity is different from the final point in the last entity. In a closed boundary these points are the same. A pocket boundary must be closed.

The POCKET command and its macro variation, DPOCK, machine an entire area within a closed boundary (comprising a string of part entities).

The tool maintains a constant tool/material relationship while milling the area to be machined. You can identify and isolate up to 20 islands comprising part entities within a pocket that the tool will avoid during machining.

For example, to clear an area with four entities defining the outer pocket boundary, enter

```
NC:> POCKET: d1 d2 d3 d4;
```

You can also select the bottom face or faces of the pocket instead of a wireframe boundary. The outside boundary of these faces will provide the pocket boundary. As in 3- and 5-axis applications, you can easily select these faces using edges, to avoid the necessity of showing a mesh.

Non-planar faces are supported in the same manner as Nsplines are supported as boundary entities. PLANE ZWORK still controls the machining level.

Faces are also selectable to define islands. The outside boundary of the island faces is used to define the island boundary.

For example, in the case of a simple cylindrical boss, the top face of the cylinder would be selected to define the island boundary. But if the sides of the boss have a draft angle, selecting the top face will not protect the base of the boss from gouging. Therefore, the boss wall will have to be selected to define the island, and

the outside (base) boundary would have to be used instead of the inside (top) boundary.

Similarly, if fillets exist around an island, the outside boundary of the fillets must be used to define the island instead of the inside boundary. This provides improved associativity between the JCF and the part geometry. Topological changes to the solid that significantly affect the boundary, but leave the bottom faces intact, will not require geometry to be reselected.

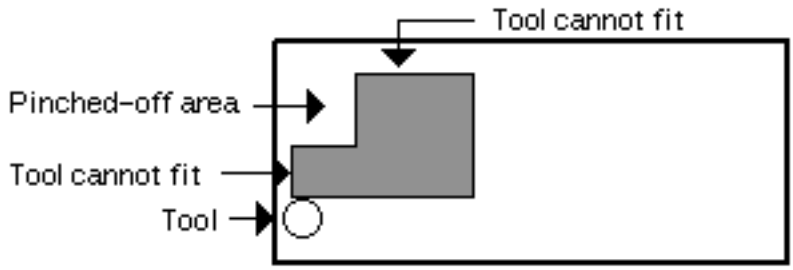
```
NC:> POCKET :Model ent ddd
NC:> POCKET FACE: edges or surfs d; ISLAND FACE: edges or surfs d;
```

You will now have no control of the order of selection of boundary entities. You can therefore use an additional modifier REV to control the direction of cut.

Pocketing a Pinched-off Area

These examples show how CVNC automatically handles pinched-off areas that cannot be reached by the tool.

A pinched-off area is a machinable region within a boundary that cannot be reached, because it is blocked by one or more areas where the tool cannot fit.

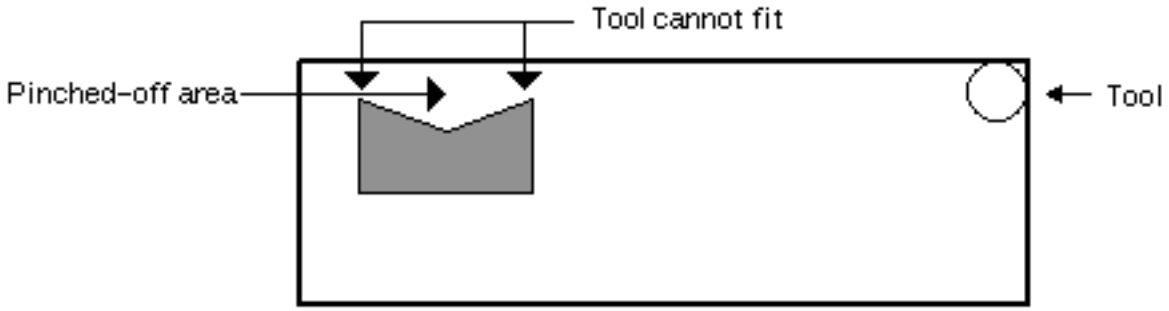


Within a pocket boundary, a single pinched-off area may be formed and still allow successful machining. Multiple pinched-off areas are not allowed.

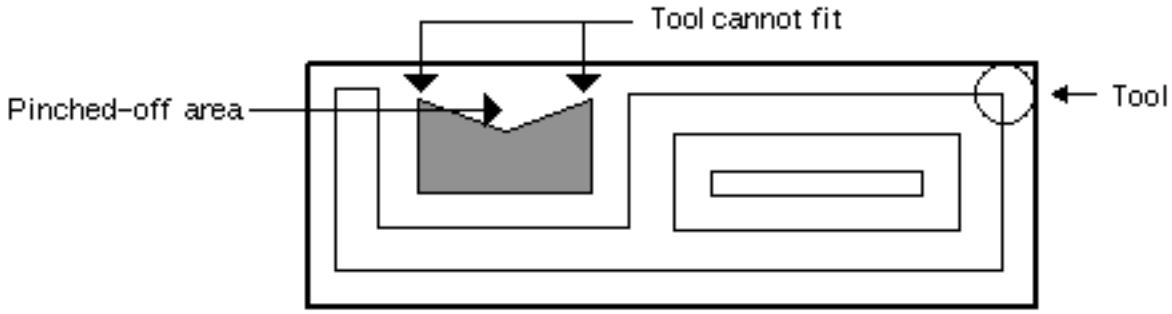
POCKET reacts differently to various situations that include pinched-off areas, as illustrated in the following examples.

Example 1

Situation: The boundary contains an island forming a single pinched-off area.

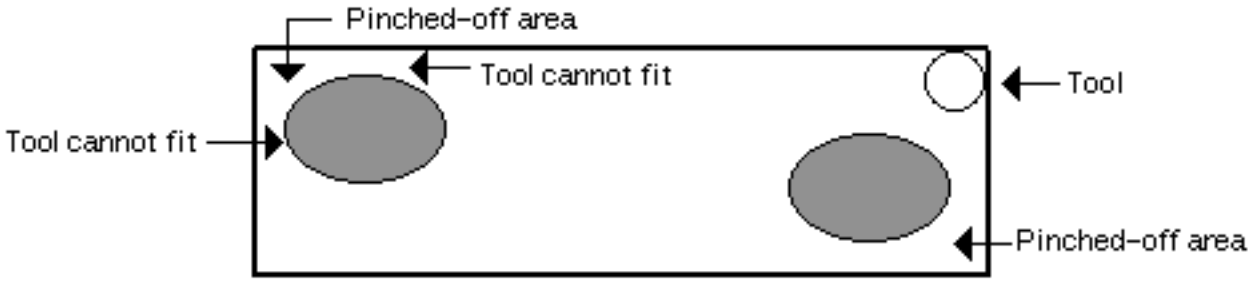


Result: POCKET detours around the island. It does not machine the pinched-off area.



Example 2

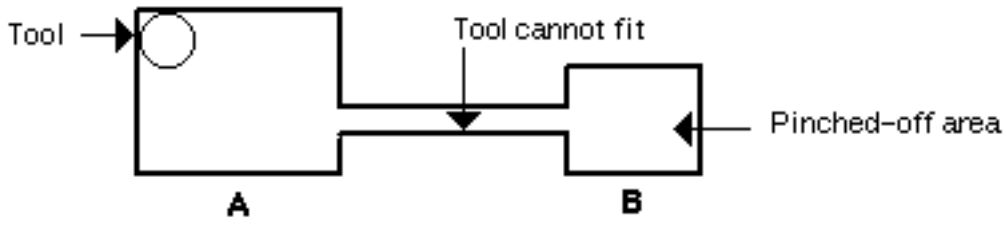
Situation: The boundary contains two islands that form two pinched-off areas. Multiple pinched-off areas are not allowed.



Result: An error message appears. POCKET does not machine the boundary.

Example 3

Situation: The boundary is divided by a thin, neck-like region where the tool cannot fit. This forms the pinched-off area A or B, depending on the current tool location. In this example, POCKET begins in area A, therefore B is a pinched-off area.



Result: POCKET machines only area A.

Pocketing with Multiple Passes — DPOCK Macro

The DPOCK macro generates multiple pocket cuts to a specified depth and at a specified angle. Use DPOCK to machine a pocket with multiple passes at different depths.

DPOCK calculates increasing material offsets at each step-down tool pass when machining an angular pocket.

For example, to machine a pocket of the NCGROUP PT99, giving it a rough tolerance of 0.0015, an incremental step-down value of 0.028 for tool movement on the z-axis, a final depth of 0.145, and a maximum stepover value of 0.16, enter

```
NC:> DPOCK PT99 RTOL .0015 ZSTEP .028 ZEND .145 MAXSTEP .16
```

For more information about macros, see *Customizing CVNC*.

Generating Area Clearance Tool Paths — AREAMILL

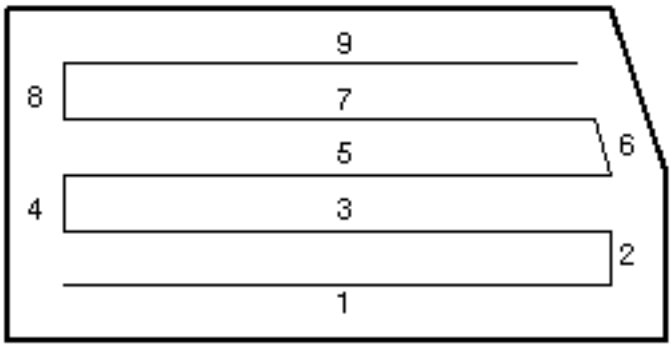
The AREAMILL (Area Milling) command generates lace cut tool paths to clear an area within a boundary and around a number of islands.

AREAMILL generates lace cut and contour tool paths to clear an area within a boundary and around islands.

AREAMILL clears an area with a succession of parallel linear cuts within the boundary of the area. The tool steps along the boundary to the next linear cut position. The next cut is parallel to the previous cut, but in the opposite direction.

The following figure shows a lace cut tool path within a boundary with no islands.

Figure 6-23 Lace Cut Tool Path with No Islands



AREAMILL, like POCKET, machines an entire area within a closed boundary and around islands. AREAMILL differs from POCKET, in that AREAMILL

- Can machine around more islands (up to 50).
- Recognizes more complex boundaries, such as overlapping boundaries.
- Can machine multiple boundaries (POCKET only allows one area).
- Switches from conventional to climb milling. (If you do not want to switch milling modes, use POCKET.)
- Automatically creates four NCGROUPs that can be referenced within a JCF. These groups are
  - %STRTPTS (Lace cutting start points)
  - %ENDPTS (Lace cutting end points)
  - %CSTARTS (Contour start points)

  - %CENDS (Contour end points)
- References start point NCGROUPs. Can reference end point NCGROUPs as a point to access a macro.
- Executes point macros. You can write CVMAC macros to be executed at predefined points during AREAMILL execution.

Please note: AREAMILL NCGROUPs and point macro predefined points are preceded by %.

For more information about macros, see *Customizing CVNC*.

AREAMILL uses variables set with DEFAMILL or default variables. AREAMILL can set new variables for use during one AREAMILL operation. At the end of that operation, the variables are restored to the values before the command.

For example, to lace cut in the direction of the x-axis at an automatically generated starting point, enter

```
NC:> AREAMILL LACE CUTDIR XAXIS BEGIN AUTO
```

As in POCKET (see “Machining Within a Closed Boundary — POCKET” on page 6-31), you can also select faces to define both the outside boundary and islands.

```
NC:> AREAMILL........ BOUND :Model ent d;
NC:> AREAMILL........ BOUND LOC: Model loc d;
NC:> AREAMILL........ BOUND FACE: edges or surfs d; ISLAND FACE:
edges or surfs d;
```

Setting Defaults for Area Clearance — DEFAMILL

Set default variables for lace cutting operations with DEFAMILL. AREAMILL can override these defaults for one operation only.

DEFAMILL sets default variables for lace cutting operations. Lace cutting is performed by the AREAMILL command.

Variables defined with DEFAMILL are retained as system variables and can be used by subsequent AREAMILL commands and other CVNC commands, such as user-defined macros.

DEFAMILL modifiers specify a starting point for lace cutting, a safe distance, contouring passes, and positioning.

AREAMILL overrides the variables set with DEFAMILL for use during one AREAMILL operation. At the end of that operation, the variables are restored to the values defined with DEFAMILL or the system default variables.

For example, to set the AREAMILL default lace cutting direction to the y-axis with a stepover value of 3% of the active tool diameter, enter

```
NC:> DEFAMILL LACE CUTDIR YAXIS STEPOVER PERCENT 3
```

Defining Boundaries and Islands — AREAMILL

User-input boundaries are processed by AREAMILL to create two types of area clearance boundaries.

When defining AREAMILL boundaries and islands, note that

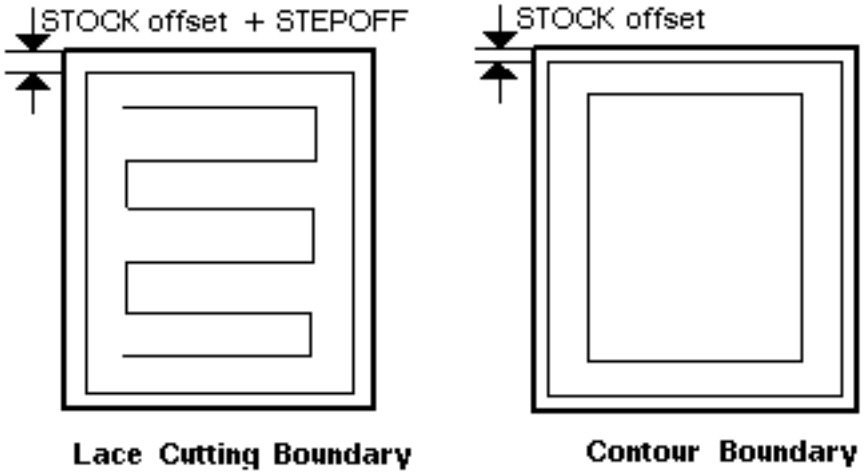
- AREAMILL supports STOCK values, either positive or negative, associated with boundaries and entities.
- AREAMILL recognizes one user-input boundary and allows up to 50 islands within the boundary.
- Each boundary may comprise up to 512 entities.
- If a boundary is open, the end points are joined with a straight line.
- A boundary must not intersect itself.
- An island may intersect with a boundary or another island, but not itself.
- Part or all of the island must lie within the boundary.

AREAMILL processes the user-input boundary to create one or both of the area clearance boundaries as shown below.

- Lace cutting boundary.

  This is the area in which the center of the tool may move during a lace cutting tool pass. Its parameters are defined by user-input contours, stock offsets, and the safe-distance offset.
- Contour boundary.

This defines the path that the center of the tool may follow during contouring moves within an initial or final contouring pass. The tool pass within the contouring boundary smooths rough edges created by the lace cutting tool pass in the lace cutting boundary. Its parameters are defined by user-input contours and stock offsets. It does not include the safe-distance offset.



Although AREAMILL is limited to one boundary, lace cutting and contouring may have more than one boundary. This occurs when an island creates a pinched-off area.

The boundary may be broken into multiple boundaries to detour around the island. Each island within a pinched-off area will be contoured before positioning to the next pinched-off area.

Performing Area Clearance Operations

There are five possible cutting operation sequences during area clearance.

During area clearance, AREAMILL cuts with any one of five operational sequences. Operations include an initial operation, initial contouring pass, lace

cutting, positioning, and final contouring pass. The following table lists operations occurring during each sequence.

Table 6-1 Area Clearance Operations

Sequence	Operation
1. Initial contouring pass	Initial operation Initial contouring pass
2. Lace cutting	Initial operation Lace cutting
3. Final contouring pass	Initial operation Final contouring pass
4. Initial contouring pass and lace cutting	Initial operation Initial contouring pass Positioning Lace cutting
5. Lace cutting and final contouring pass	Initial operation Lace cutting Positioning Final contouring pass

Initial Operation

In an initial operation, if the current tool location is on the work plane but not equal to the required location, a DIRECT move to the required location is made at CONNECT feed rate. This is a side-approach move.

During a side-approach move, the tool position before you enter AREAMILL affects the start position you select with the AUTO modifier.

When you use AUTO (the default) to select the start position of the operations, the side-approach avoids islands and moves to the start position. Note that it does not avoid islands that merge with the boundary or another island.

When you use the START modifier to select the start position of the operation, the side-approach may violate the boundary or islands depending on the tool position. In that case, a warning message is issued.

In all other cases of the initial operation besides side-approach moves, a standard initial plunge is made. This plunge consists of the following moves:

1. Retract to CLEAR plane at CLEAR feed rate.
2. Move on CLEAR plane at CLEAR feed rate.
3. Plunge to APPROACH plane at APPROACH feed rate.
4. Plunge to WORK plane at PLUNGE feed rate.

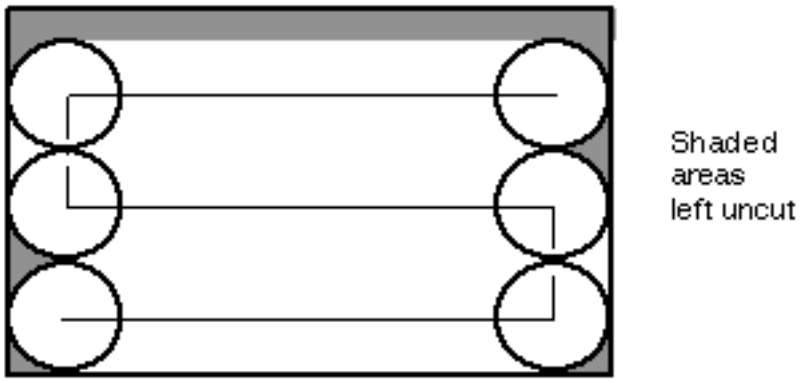
Initial and Final Contouring Passes

Both initial and final contouring passes consist of contouring operations combined with positioning moves. The initial pass is done at CONNECT feed rate; the final pass is done at CUT feed rate. If the stepoff value is equal to zero, there are no positioning moves.

Please note: If lace cutting and a final contouring passes are specified without an initial contouring pass, the remaining uncut material on the part after the lace cut (see the following figure, Uncut Material Left by a Lace Cut) may be gouged when (A) the tool repositions for contouring, and (B) the tool positions between islands during contouring.

Figure 6-24 Uncut Material Left by a Lace Cut

NC:> **AREAMILL LACE CONTOUR FINAL BOUND:** MODEL Loc **$d_1 d_2 d_3 d_4$**



Lace Cutting

Lace cutting alternates cutting and stepping moves, combined with the necessary positioning moves. Cutting moves are at the CUT feed rate, except for the first boundary cut created by a pinched-off area, which is at CONNECT feed rate. If an initial contouring pass has been made, the first feed rate cut will be at CUT feed rate. All stepping moves are made at CONNECT feed rate.

Positioning

Positioning may be required

- During an initial contouring pass
- Between an initial contouring pass and lace cutting
- Within lace cutting
- Between lace cutting and a final contouring pass
- During a final contouring pass

The following figure, AREAMILL Command with the RETRACT Modifier, is a step-by-step illustration of RETRACT positioning in a lace cut tool path within a boundary containing islands. When positioning, the tool retracts to the ZRETRACT plane.

The example in this figure shows you how to use the AREAMILL command with the RETRACT modifier.

Figure 6-25 AREAMILL Command with the RETRACT Modifier

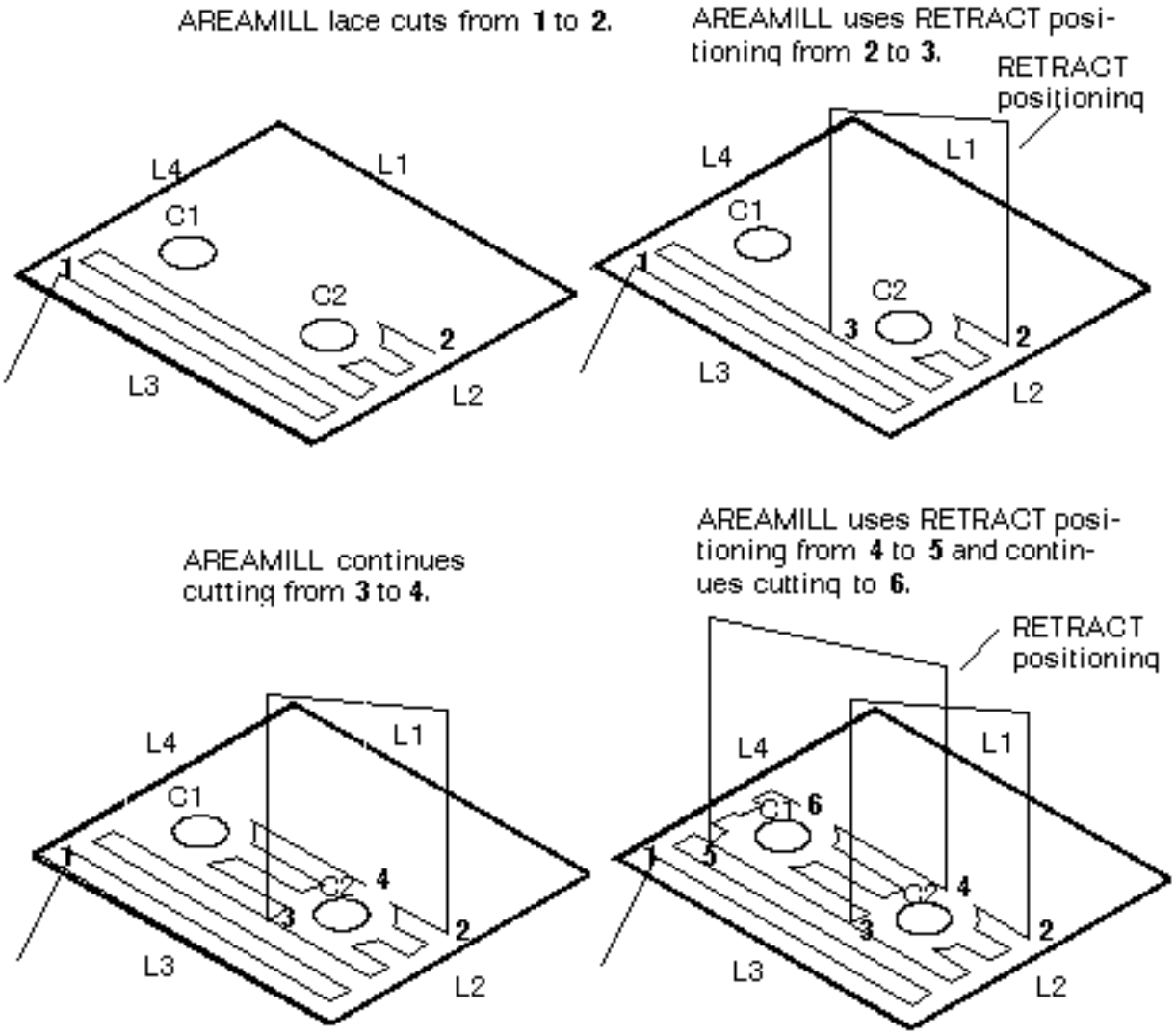
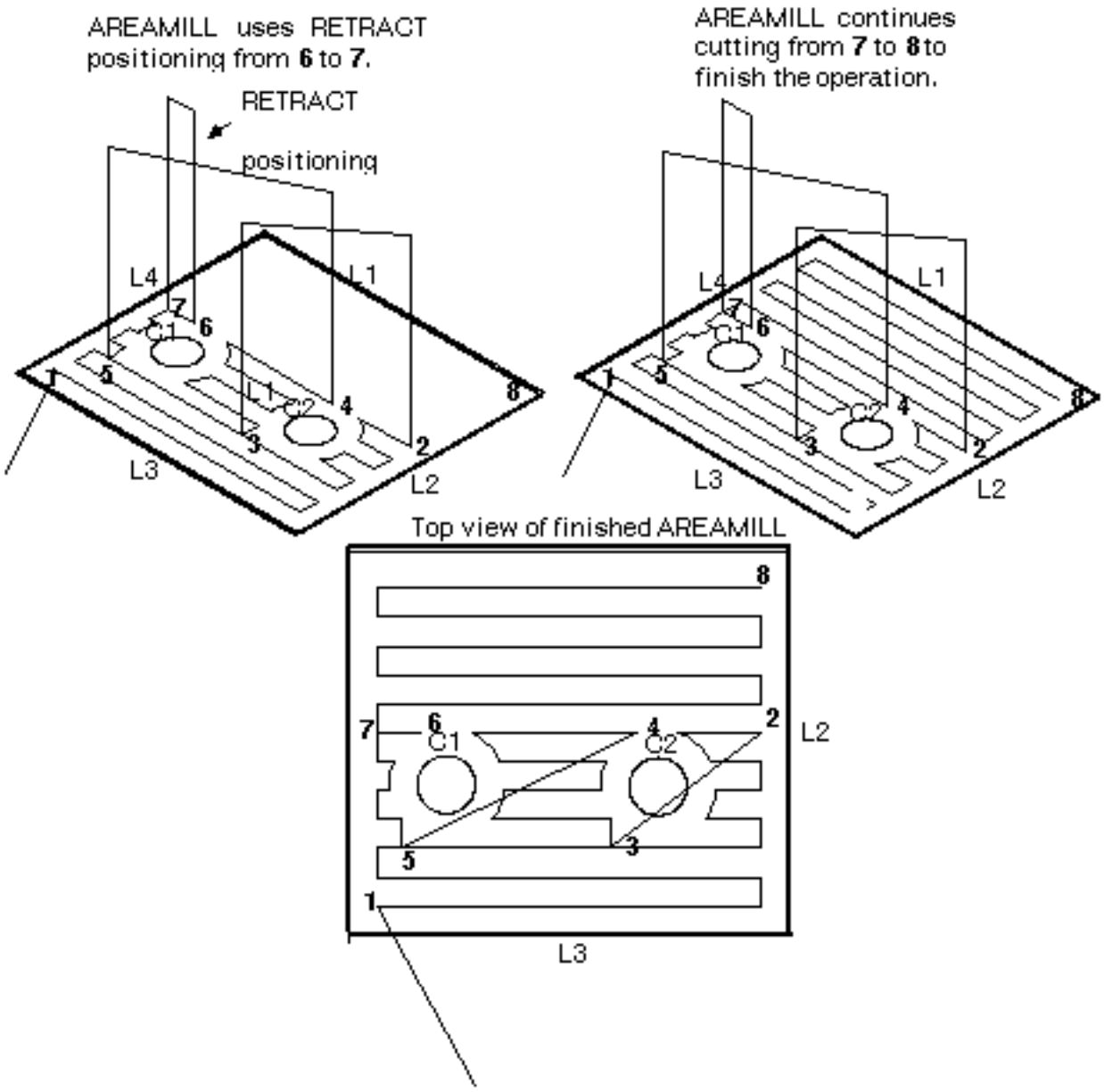


Figure 6-26 AREAMILL Command with the RETRACT Modifier, (continued)



The QUERY modifier is available to generate and write all tool path information to system variables without generating a tool path.

Adding Machine Control Statements

Use point macros (macros referenced at predefined points) to add machine control statements during tool path generation.

AREAMILL can reference CVMAC point macros at predefined points during its execution.

Point macros, by passing text to the output, add machine control statements during AREAMILL tool path generation. This text is in addition to the standard command output.

If a point macro is present in the active macro libraries, it is executed every time that point is reached; if a point macro is not present, no action is taken.

A list of all the predefined points is provided in the *CVNC Milling Command Reference,* Appendix C, CVNC Point Macro Points.

For more information about macros, see *Customizing CVNC.*

Using AREAMILL Point Macros

To use an AREAMILL point macro, you must write a CVMAC macro and give it the same name as the predefined point at which you want to invoke the macro. For example, amil05 is the macro for the predefined point AMIL05.

Please note: The point macro name must be in lowercase or CVNC cannot read or locate it. CVMAC can perform a variety of activities, such as

- Eliciting input from the operator
- Reading or writing from files
- Reading or writing from the CADDS database
- Performing algebraic calculations

Please note: The following are not allowed in AREAMILL point macros:

- CVNC system commands
- CVMAC geometry commands
- Macro calls and command files
- Arguments

For more information on CVMAC, see *CVMAC Language Reference.*

After you write the macro, install it in the proper libraries. To do this

1. Create a source file in the CVMAC language using an OS editor.
2. Compile this macro using the OS `cvmcomp` command.
3. Create a link file using an OS editor. This link file lists a set of macros to be linked in a single CVMAC executable file. It also provides a name for that executable file.
4. Enter the OS `cvmlink` command to create a single CVMAC executable file from the set of macros listed in the link file.
5. Enter MACLIB within CVNC to identify the CVMAC executable file you created containing your macros.

After you have written and installed the macros, and have entered MACLIB, CVNC will automatically access the macros when the predefined point is reached.

For more information on building point macros, see *Customizing CVNC*.

Generating Output with AREAMILL Point Macros

To generate output, point macros must contain either CVNC pass-through statements or TEXTOUT statements. For more information on pass-through statements and macros, see *Customizing CVNC*; for more information on TEXTOUT, see *CVNC System User Guide and Menu Reference*.

Using Variables

You can assign variables to each predefined point. With these variables, you can hold the text to output at each point.

Use ASSIGN to define or change variables from within your JCF; you do not have to alter the macro to change the output text. For example, the following commands:

```
NC:> ASSIGN OUTPUT %AMIL05 'SPINDL 2000 RPM'
NC:> AREAMILL BOUND $L1 $L2 $L3 $L4;
```

make an internal call to the amil05 macro

```
PROC amil05
DECLARE TEXT a
APLSYSR/"%AMIL05",a
!TEXTOUT "{a}"
```

and insert the text string ‘SPINDL 2000 RPM’ into the output at the point AMIL05.

For more information on ASSIGN, see *CVNC System User Guide and Menu Reference* and *CVNC Milling Command Reference.*

Example of an AREAMILL Point Macro

The following JCF and macro inserts the statements ‘SPINDL 2000 RPM’ and ‘COOLNT MIST’ before every DIRECT positioning move in the output.

```
NC:> ASSIGN OUTPUT %AMIL05 ‘SPINDL 2000 RPM ! COOLNT MIST’
NC:>AREAMILL BOUND $L1 $L2 $L3 $L4;
<#
<# This macro converts a ! delimited string in %AMIL05, passed
<# from CVNC, into separate pass-through output statements
<#
      PROC amil05
      DECLARE TEXT &OUTSTR (255)
      DECLARE REAL A, B
<#
<#  Get a string from CVNC and add a final !delimiter;
<#initialize substring pointers A and B to the first substring
<#
      APLSYSR/”%AMIL05”,&OUTSTR
      A = 1
      &OUTSTR = &OUTSTR + “!”
      B = FNDB(“!”,&OUTSTR)
      WHILE B > A
<#
<#  Output string
<#
      !TEXTOUT
“{&OUTSTR(A,B)}”
<# Set A and B to the next substring
<# Note that B > A only if a valid substring is found
<#          A = FNDA
(“!”,&OUTSTR(B,))+B-1
            B = FNDB
(“!”,&OUTSTR(A,))+A-1
      ENDWHILE
      RETURN
      END
```

Chapter 7

Tool Motion Generation: Hole Processing

CVNC-M2 hole processing commands perform drilling (DRILL), boring (BORE), countersinking (CSINK), and tapping (TAP) operations. These are cycle commands: they can perform their function once or many times in an operation.

- Overview of Hole Processing Commands
- Drilling — DRILL
- Boring — BORE
- Countersinking and Chamfering — CSINK
- Tapping — TAP
- Controlling Hole Depth
- Controlling Clearance Distances
- Identifying Locations and Order of Machining
- Setting Dwell Time
- Setting Avoidance Parameters
- Hole Processing on a Cylindrical Part
- Displaying Machine Tool Motion

Overview of Hole Processing Commands

Use the CVNC-M2 hole processing commands to perform

- Drilling (DRILL)
- Boring (BORE)
- Countersinking (CSINK)
- Tapping (TAP)

With hole processing command modifiers, you can

- Control hole depth
- Control clearance distances
- Identify location and order of machining
- Specify methods for hole processing
- Set dwell time
- Set avoidance parameters

Hole Processing Methods

You can use the hole processing commands in two ways:

Method 1:
Set all parameters in the first command line in your JCF. Then, use individual command lines to specify the hole locations.

Method 2:
Set parameters and locations in each command line.

If your operation requires a number of holes in different locations to be drilled with the same depth and dwell, for example, Method 1 may be an efficient choice.

If your operation specifies multiple holes at varying parameters (such as depth), use Method 2.

Please note: Changing any parameters on a command line resets all others (except SAFDIST and ENDDIST) to the CVNC defaults.

Method 1

In this method, use the first hole processing command line in your JCF to specify all the parameters, except the hole locations. Then, use subsequent command lines to specify the hole locations. All of the parameters previously set by you (including any system defaults) remain in effect throughout the operation. For example,

```
NC:> DRILL DEPTH 1.0 SAFDIST .1
NC:> DRILL : Model loc d1 d2
NC:> DRILL : Model loc d1
```

In this example, three holes are drilled. All three are to a depth of 1, and have a clearance distance of .1.

Method 2

In this method, set all the hole parameters and specify the hole locations in the command line. For example,

```
NC:> DRILL DEPTH 2.0 SAFDIST .1 : Model loc d1
NC:> DRILL DEPTH 2.0 SAFDIST .2 : Model loc d1
NC:> DRILL DEPTH 2.0 SAFDIST .3 : Model loc d1
```

In this example, three holes are drilled. All three have separate clearance heights.

(If the SAFDIST value remains the same for each DRILL, that modifier does not need to be respecified, as it is a modal parameter. Modal parameters retain the most recent setting within the JCF. An ENDDIST value for THRU holes is also a modal setting.)

Please note: The IN and INX [ED] commands cannot be applied to hole processing commands. Hole processing commands cannot be decomposed into equivalent sequences of lower-level JCF commands.

Displaying Tool Motion

Use DISPLAY CYCLE [SYS] to generate a graphic display of machine tool cycles that result from hole processing operations.

Drilling — DRILL

The DRILL command enables you to drill holes in various modes.

Use DRILL to drill blind (to depth) and thru-holes in standard, peck, and break-chip mode.

With DRILL, you can specify an incremental distance or a z-location for drilling to depth; the same is true for drilling a thru-hole. You can also specify a clearance distance used for approach and retract.

For thru-holes, you can modify the depth value of drilling motion by a specified increment.

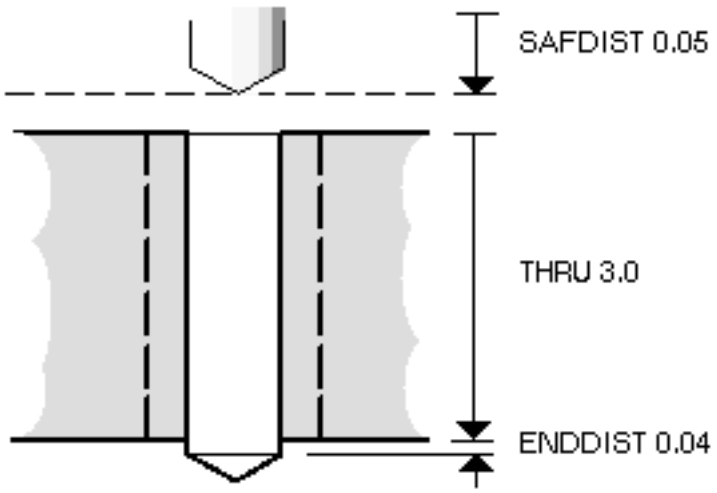
Other DRILL modifiers enable you to

- Select the vertical face or faces of a solid
- Determine clearance above a location
- Determine the DEPTH distance
- Specify the amount of dwell in seconds or revolutions
- Specify avoidance parameters

To drill a typical thru-hole, enter DRILL, as shown in the following example.

```
NC:> DRILL THRU 3.0 SAFDIST .05 ENDDIST .04
```

Figure 7-1 Drilling with DRILL



Specifying Drilling Methods

In addition to the standard drilling method, you can choose one of two other drilling methods. You also have a choice of several methods to control the depth of tool passes.

You can perform drilling in the standard way or choose one of these methods:

- PECK (deep-hole drilling)
- BREAK-chip (pause drilling)

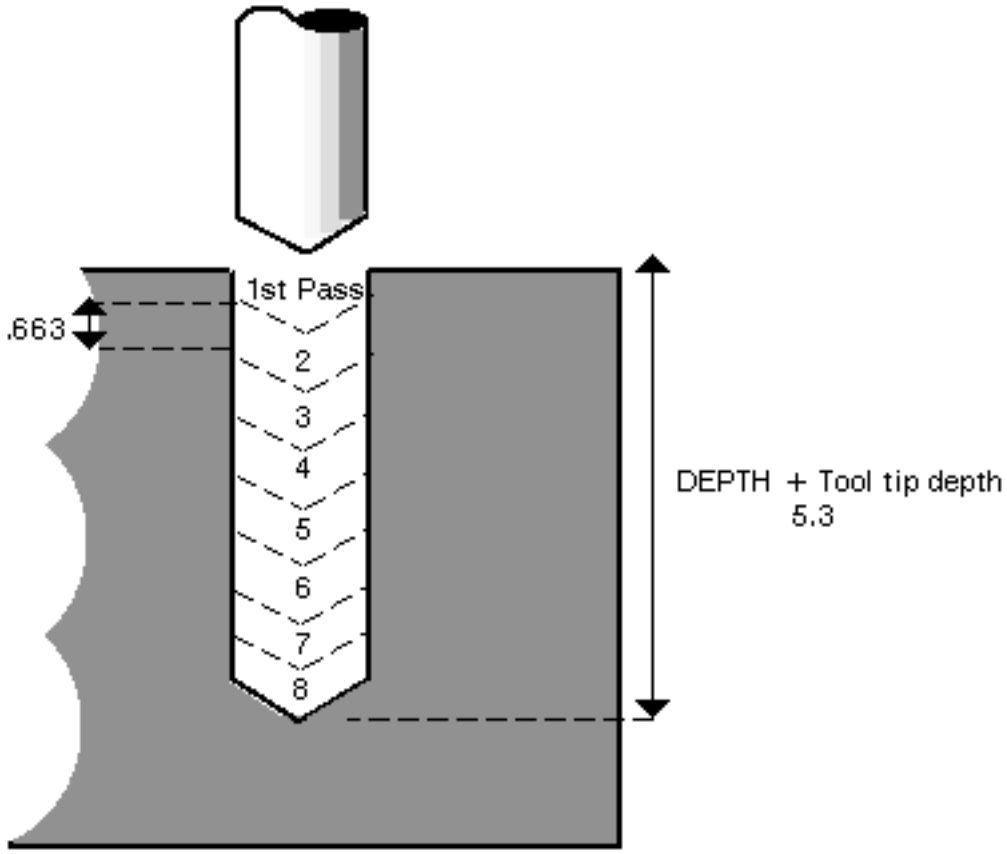
Between passes, PECK retracts the tool from the hole to the clearance distance. BREAK causes the tool to pause between passes. With peck or break-chip drilling, you must set parameters for tool passes.

MAXDEP specifies a maximum cut depth for each tool pass. CVNC calculates the necessary number of passes, with cut depths equal to or less than MAXDEP. All passes cut equal depths.

The following figure shows how to use DRILL for a MAXDEP of 0.75. CVNC calculates that eight tool passes are necessary to cut the specified depth with equal passes that do not exceed MAXDEP. As seen in this figure, each pass is .663.

```
NC:> DRILL DEPTH 5.0 PECK MAXDEP 0.75
```

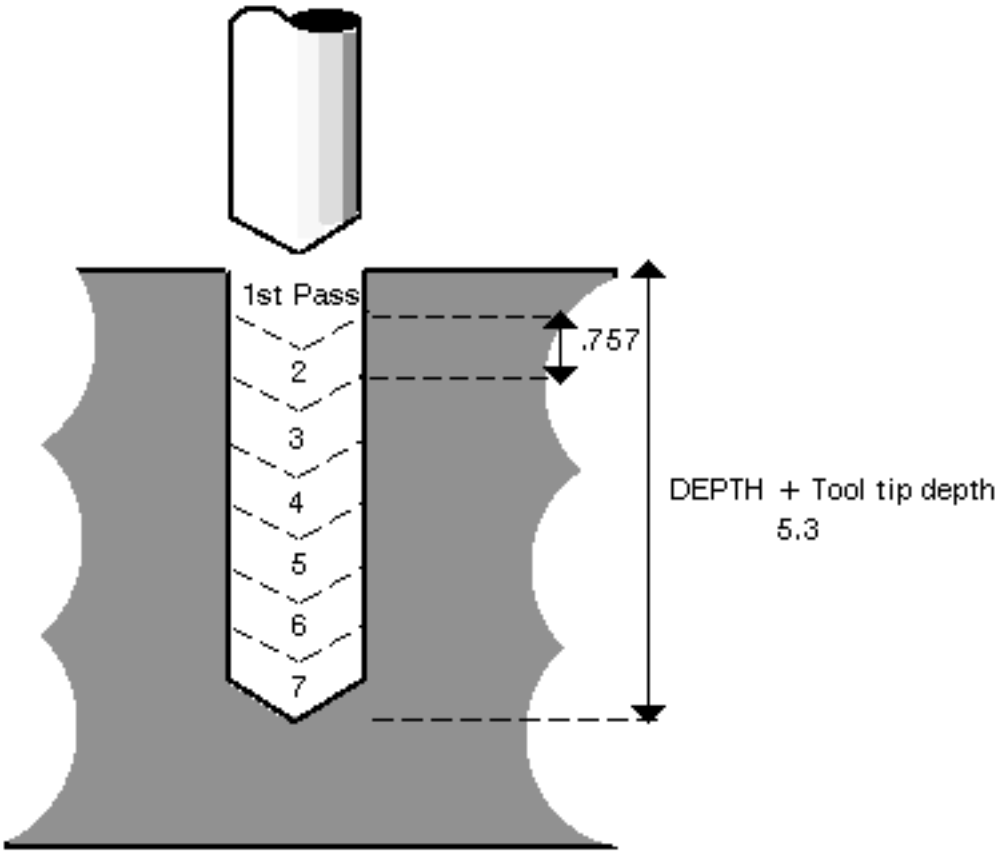
Figure 7-2 Drilling with MAXDEP



Alternatively, you can use NPASS. NPASS specifies the number of tool passes, and CVNC calculates an equal depth for each pass, as illustrated in the following figure.

```
NC:> DRILL DEPTH 5.0 BREAK NPASS 7
```

Figure 7-3 Drilling with NPASS

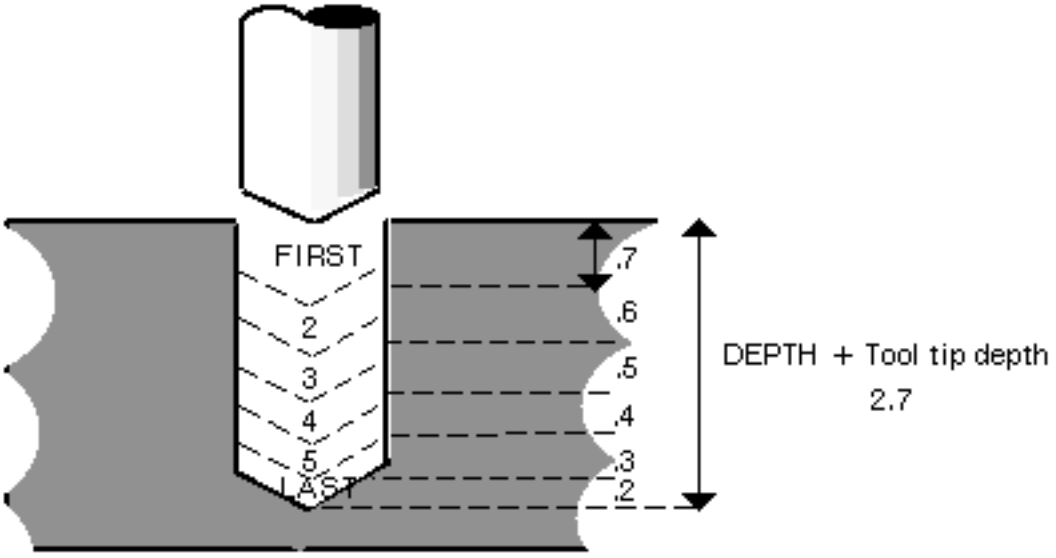


FIRST and LAST are used to specify the depth of the first and last tool passes. The FIRST pass must be of a greater depth than the LAST. CVNC calculates linearly decreasing depths for each intermediate pass, as shown in the following figure.

To use this method of controlling tool passes, follow this example below.

```
NC:> DRILL DEPTH 2.5 PECK FIRST 0.7 LAST 0.2
```

Figure 7-4 Drilling with FIRST/LAST



Boring — BORE

The BORE command often follows DRILL, making the hole more precise. Use BORE to bore holes to specified depths or through surfaces in standard, manual, and automatic modes.

As in DRILL, and TAP, you can also use the FACE modifier to select the vertical face or faces of a solid.

You can specify a clearance distance to be used for approach and retraction or use the default (0.1 inch or 2.54 mm). You can incrementally modify the depth of a drilling position by a specified value. Clearance distance (SAFDIST) and depth of drilling motion (ENDIST) are modal parameters that need not be respecified.

As an alternative to SAFDIST, use the ZPLANE modifier to

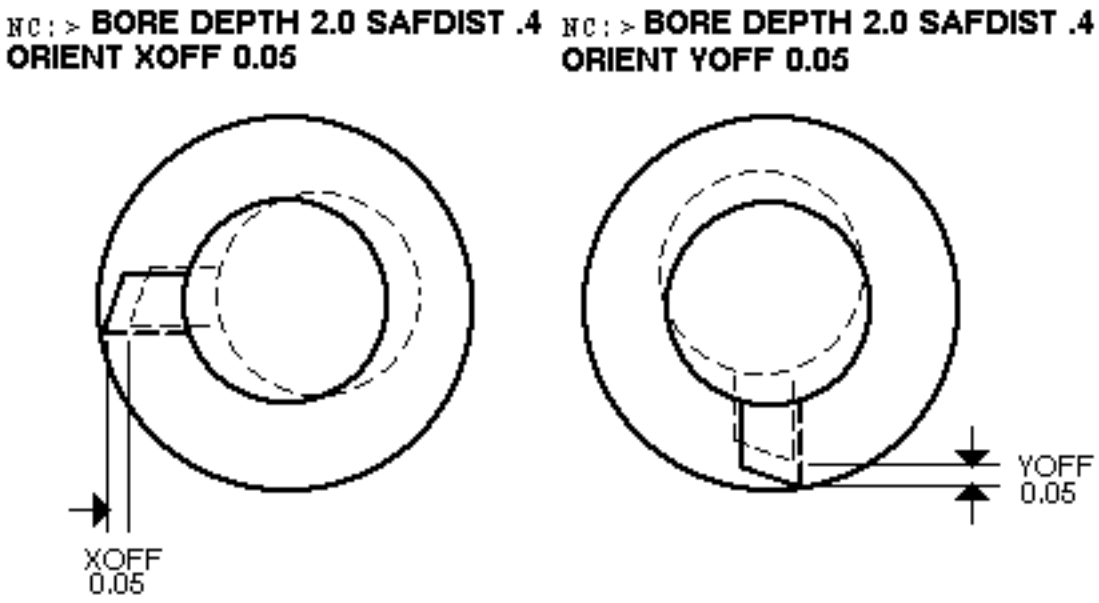
- Invoke ZAPPR and ZRETRACT (defining approach and retract points), if specified with DEPTH or THRU
- Invoke ZWORK (defining the bottom of the hole), if specified instead of DEPTH or THRU

Please note: When using BORE, always specify positive depth (with the DEPTH modifiers) and thru-hole (with the THRU modifiers) values.

Using the MANUAL modifier, you can stop the tool at the end of a boring stroke so the machine operator can manually adjust the tool away from the surface to prepare for retraction.

Other modifiers specify dwell time or revolutions of dwell, orient the tool and move it away from the surface, and specify boring locations and avoidance parameters.

To use BORE, follow these examples:



Specifying Boring Methods

There are three options for controlling the tool following a boring operation. When using BORE, you can choose from these three methods:

- MANUAL
- ORIENT
- Default

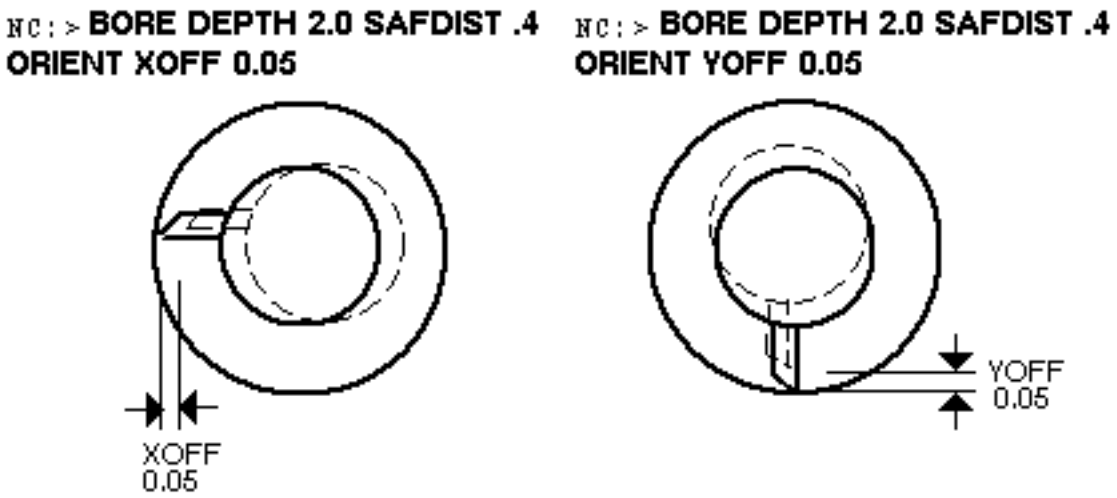
MANUAL stops the tool at the end of a boring stroke, so that the machine operator can manually adjust the tool away from the surface to prepare for retraction.

ORIENT automatically orients the tool along the x- or y-axis and moves it away from the surface of the hole by an incremental value (specified with XOFF exp or YOFF exp) before retraction.

The default method bores in and out of the hole.

The following examples show how to use the ORIENT modifier with BORE.

Figure 7-5 Using ORIENT with BORE



Countersinking and Chamfering — CSINK

Use the CSINK command for your countersinking and chamfering operations, working to a specified diameter. Once you have indicated the diameter to be chamfered, use this value with tool parameters to calculate the depth of tool motion.

You can specify a clearance distance for approach and retraction or use the default (0.1 inch or 2.54 mm). You may incrementally modify the depth of a drilling position by a specified value. Clearance distance (SAFDIST) and depth of drilling motion (ENDIST) are modal parameters that need not be respecified.

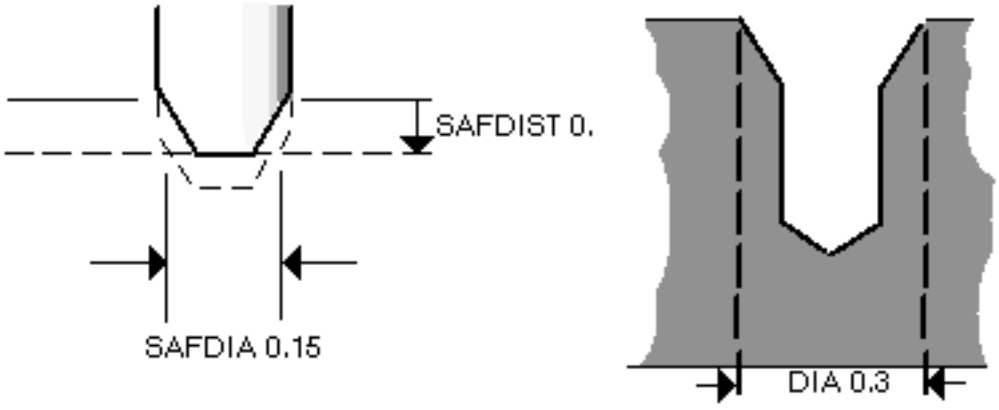
In addition to the clearance distance, you can set a clearance diameter and seconds or revolutions of dwell.

Specify points or arcs (from which the arc center is derived) that are chamfering locations. You can also set CSINK to machine a list of locations or entities in reverse order.

When you establish avoidance/clearance parameters, you can use absolute or incremental z-values for the tool retraction.

For example, enter this command to produce the results in the following illustration:

```
NC:> CSINK DIA 0.3 SAFDIST 0.1 SAFDIA 0.15
```



Tapping — TAP

Use the TAP command for tapping blind and thru-holes.

You can specify an incremental distance for tapping to depth or a thru-hole. In either case, this value is subtracted from the z-coordinate of the locations to be drilled.

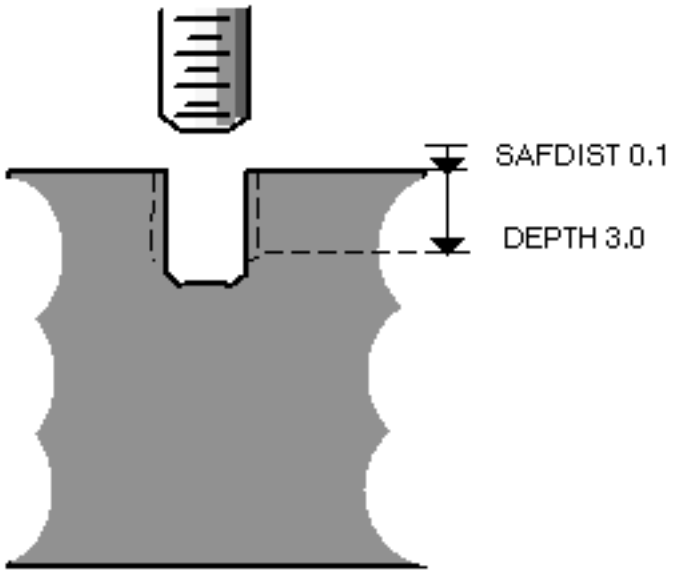
As an alternative, you may specify a z-location for tapping to depth or for tapping a thru-hole. A second alternative is to specify an entity from which the z-origin value is obtained for tapping to depth or tapping a thru-hole. (Positive depth and thru-hole values must be used.) As in DRILL, and BORE, you can also use the FACE modifier to select the vertical face or faces of a solid.

Set clearance distance for tool approach and retraction with the SAFDIST modifier or by using the ZPLANE modifier. The ZPLANE modifier invokes ZAPPR and ZRETRACT planes to determine clearance above a location. When used instead of DEPTH or THRU, ZPLANE invokes the use of ZWORK plane to determine the DEPTH distance.

With the ENDDIST modifier, you can modify the depth value of tapping motion by a specified increment. This value is subtracted from the depth you specified. The ENDDIST value is modal for all thru-holes and need not be respecified.

Set tapping locations and, optionally, a list of locations or entities to be machined in reverse order. For example,

```
NC:> TAP DEPTH 3.0 SAFDIST .1
```



Controlling Hole Depth

You can control hole depth during a BORE, DRILL, or TAP operation with the DEPTH, THRU, and ENDDIST modifiers. You can control hole depth during a CSINK operation with the DIA modifier.

Controlling Hole Depth During a BORE, DRILL, or TAP

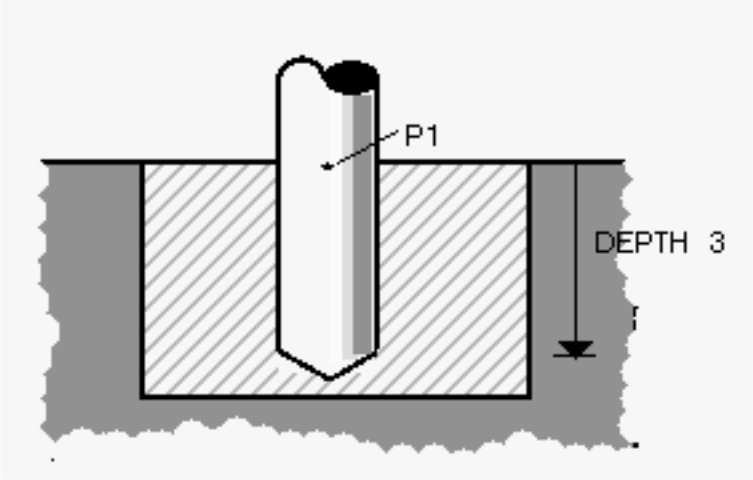
With BORE, DRILL, and TAP you can machine to a depth specified with the DEPTH modifier or you can machine a thru-hole with the THRU modifier.

The DEPTH value specifies the depth of the full diameter hole. CVNC compensates for the drill point.

The following figure shows how to drill a blind hole with CVNC, where $P1 is the top of the hole.

Figure 7-6 Drill Depth: Blind Hole

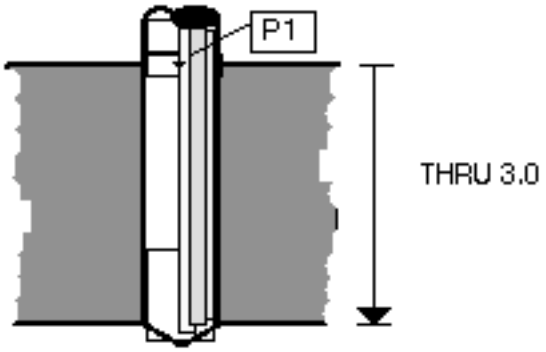
```
NC:> DRILL DEPTH 3.0 ENT $P1
```



The following figure shows how to drill a thru-hole.

Figure 7-7 Drill Depth: Thru-Hole

```
NC:> DRILL THRU 3.0 ENT $P1
```

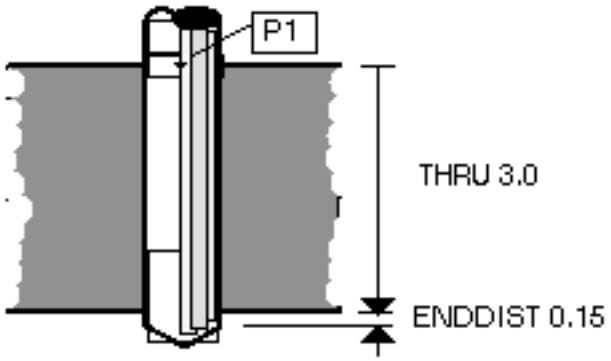


When drilling thru-holes, use the ENDDIST modifier to ensure that the full diameter of the tool clears the underside of the wall. ENDDIST adds its specified value to the THRU value.

The following figure shows how to use the ENDDIST modifier when drilling a thru-hole.

Figure 7-8 Using ENDDIST when Drilling a Thru-Hole

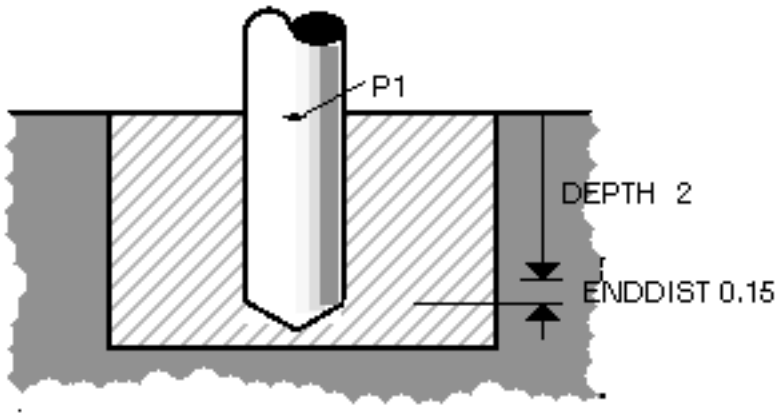
```
NC:> DRILL THRU 3.0 ENDDIST 0.15 ENT $P1
```



When the drilled hole is to be tapped, you may want to drill deeper than the threaded portion. The following figure shows how to use the ENDDIST modifier to extend the depth of a blind hole.

Figure 7-9 Using ENDDIST to Extend the Depth of a Hole

```
NC:> DRILL DEPTH 2.0 ENDDIST 0.15 ENT $P1
```



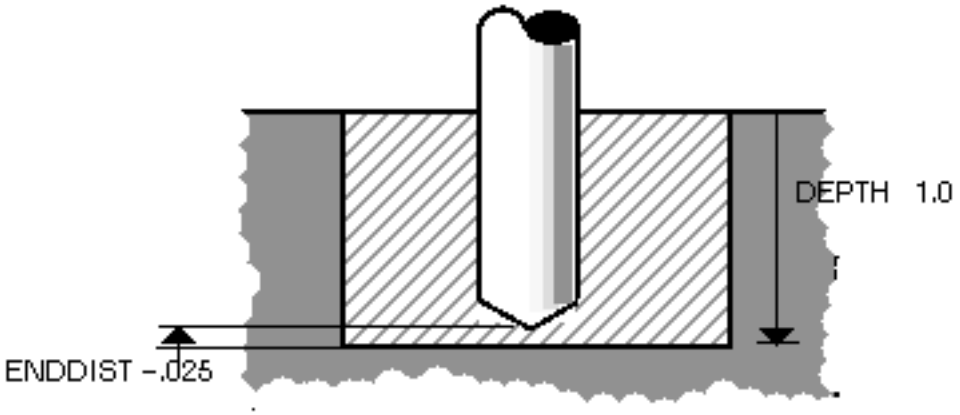
When machining DEPTH holes, CVNC adds only an ENDDIST value when you use the ENDDIST modifier. This value is stored in the #DEPCLR system variable.

DEPTH and THRU distances apply to the full diameter of the tool. When drilling, however, you can use the NOANGL modifier so that the tool point angle value is not used when calculating the drill depth.

The following figure shows you how to use DRILL with ENDDIST and NOANGL to drill a hole in preparation for a pocketing operation. By using a negative ENDDIST value and NOANGL, DRILL removes material without gouging the bottom of the pocket.

Figure 7-10 Using the ENDDIST and NOANGL Modifiers with DEPTH

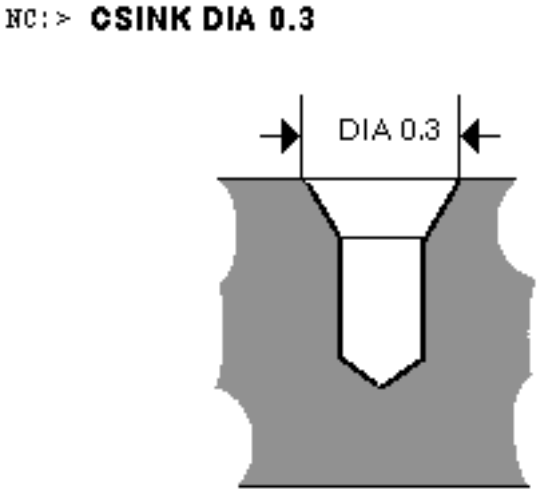
```
NC:> DRILL DEPTH 1.0 ENDDIST -.025 NOANGL
```



Controlling Hole Depth During a CSINK

CSINK machines to a specified depth. This depth is calculated from a specified arc (DIA ent) or from the diameter to be chamfered (DIA exp). The latter is shown in the following figure.

Figure 7-11 Using the DIA Modifier with CSINK



Controlling Clearance Distances

With the hole processing commands, you can control clearance distances with the SAFDIST or ZPLANE modifiers.

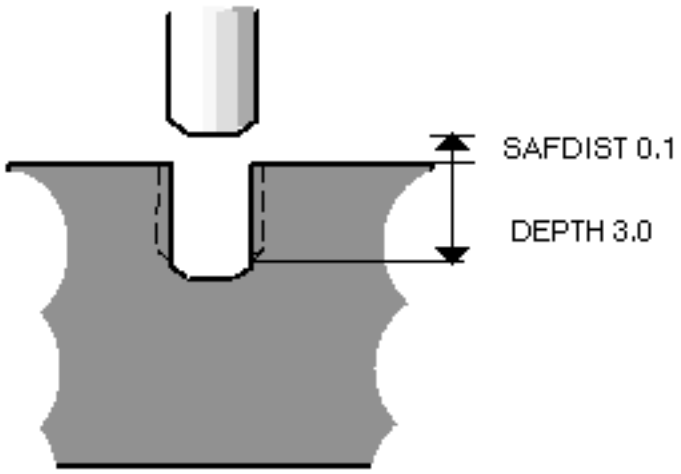
SAFDIST specifies a value to be added to the z-value of the hole location, as shown in the following figure. This value is used for approaches and retractions.

ZPLANE causes the values of the ZAPPR and ZRETRACT planes to determine clearance. When used instead of DEPTH or THRU, ZPLANE causes the value of ZWORK to be used for the depth.

The following example shows how to use the SAFDIST modifier with the TAP command:

```
NC:> TAP DEPTH 3.0 SAFDIST .1
```

Figure 7-12 Controlling Clearance with SAFDIST

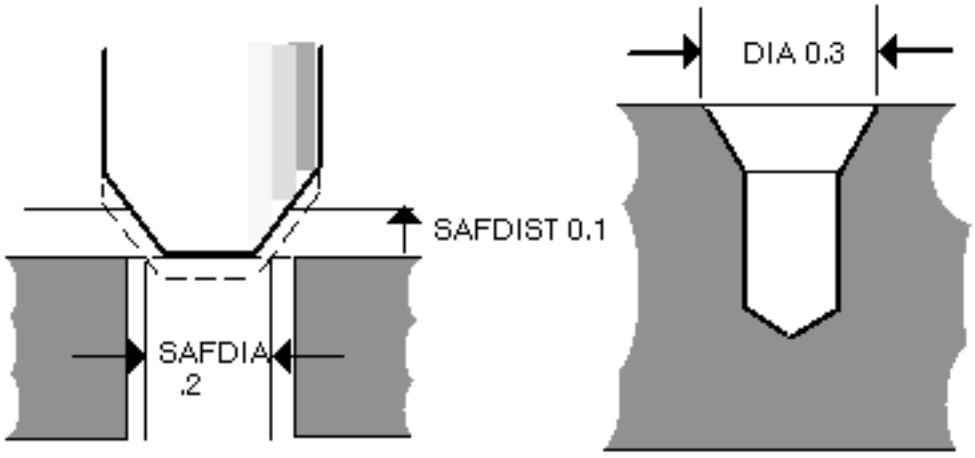


In addition to a clearance distance, CSINK also allows you to specify a clearance diameter with the SAFDIA modifier. This value is used with the tool parameters to calculate a subsequent z-value for clearance after the tool has approached the hole location, as illustrated in the following figure.

This example shows how to use the SAFDIST and SAFDIA modifiers with the CSINK command:

```
NC:> CSINK DIA 0.3 SAFDIST 0.1 SAFDIA 0.2
```

Figure 7-13 Controlling Clearance with SAFDIA



Identifying Locations and Order of Machining

You can reverse the usual milling order or use the NCGROUP command and the OPTIM modifier to minimize distance traveled in a hole processing operation.

You can identify the location of a hole-processing operation with coordinate locations (locs) or entities (ENT ents). You can also use NCGROUPs. Machining occurs in the order these locations are entered, unless you do one of the following:

- Use the REV modifier to reverse the order of machining.
- Specify an NCGROUP (in place of locs or ents) that was defined with the OPTIM modifier.

The REV modifier is available with all the hole processing commands.

See the *CVNC System User Guide and Menu Reference* for more information about NCGROUPs.

Setting Dwell Time

You can set dwell time in two different ways for the DRILL, BORE, and CSINK commands.

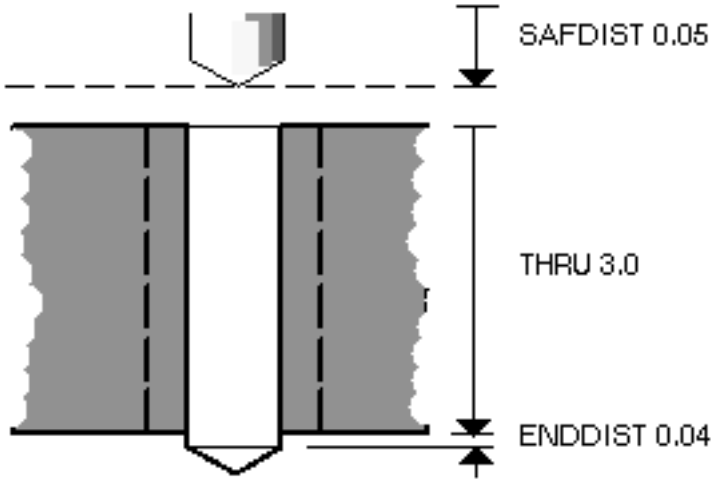
With DRILL, BORE, and CSINK, you can specify the amount of dwell time the tool has after it has reached the specified depth. You can use either of these modifiers to do this:

DWELL exp, to specify dwell time in seconds

REVS exp, to specify the number of revolutions of dwell

For example, in a drilling operation such as the following one, set a dwell time of 2 seconds by entering

```
NC:> DRILL THRU 3.0 SAFDIST .05 ENDDIST .04 DWELL 2
```



Setting Avoidance Parameters

Use the AVOID modifier with the DRILL, BORE, CSINK, or TAP commands to specify a secondary z-clearance for the hole processing commands.

You can use the AVOID modifier with all hole processing commands to specify a secondary z-clearance for a subgroup of locations (locs) or entities (ENT ents).

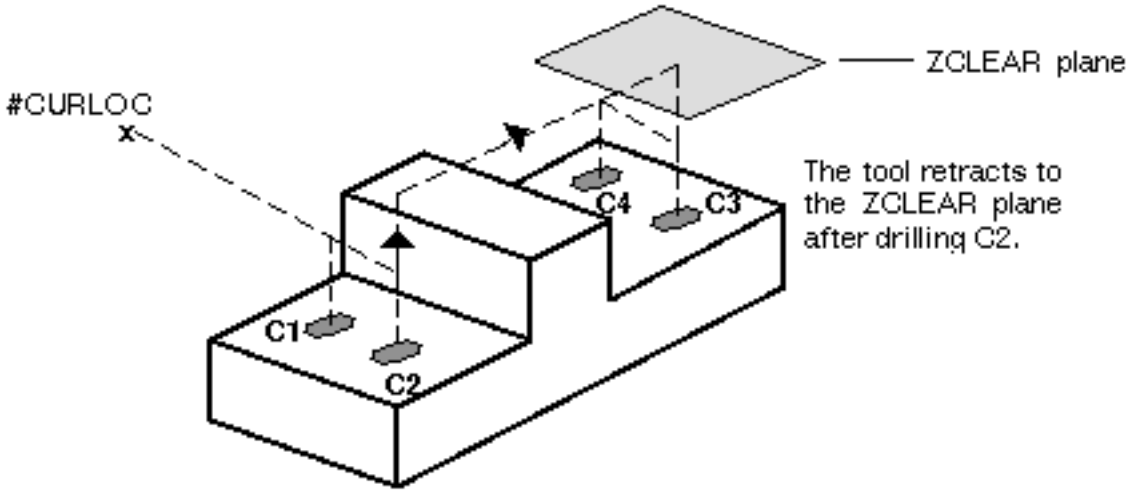
AVOID and subsequent modifiers cause the tool to retract to one of the following after machining the hole location(s) specified with AVOID:

- ZCLEAR plane (the default)
- Absolute value of z (ZABS exp) before positioning to the next hole location
- Incremental value of z (ZINC exp) added to the normal retraction z-value

The following example shows how to use AVOID to retract to the ZCLEAR plane after drilling $C2.

Figure 7-14 Using the AVOID Modifier

```
NC:> DRILL ENT $C1 $C2 $C3 $C4; AVOID ENT $C2; ZCLEAR
```



Hole Processing on a Cylindrical Part

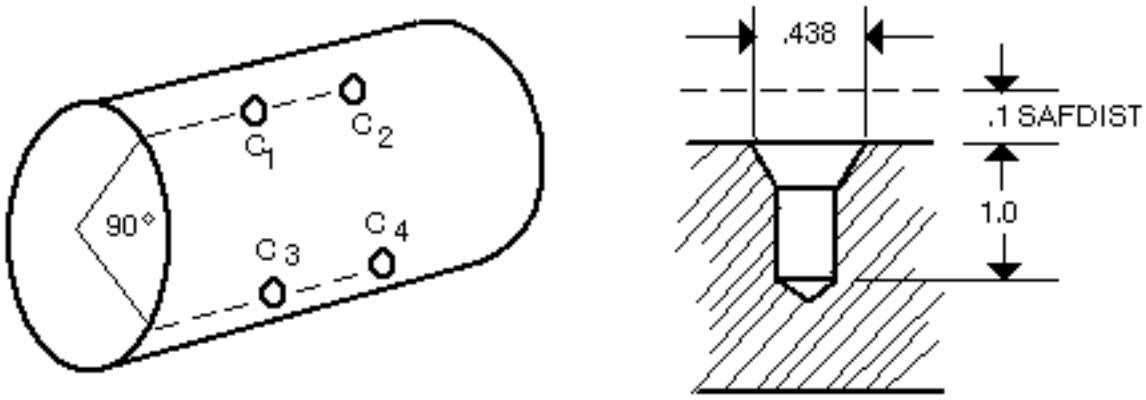
When machining a cylindrical part, you can use INDEX to invoke rotary table motion from the current position around one or two pivot points with respect to the to the x-, y-, and z-axes. (You must specify an axis already defined with CONFIG.) Specify the new position of the tool with an absolute or incremental value of the angle of rotation.

INDEX generates and selects a new CADDS Cplane accordingly. (See Chapter 4 for more information on INDEX.)

To index a rotary axis, follow the next example. The sample JCF, and the following figure, show two hole processing commands used to machine a cylindrical part.

```
NC:> DEFTOOL "DRILL" DRILL DIA .25 ATANGL 118
NC:> DEFTOOL "CSINK" CSINK DIA .5 BETA 45 FLAT .125
NC:> INDEX BAXIS ATANGL 0
NC:> CHGTOOL 1 "DRILL"
NC:> DRILL DEPTH 1.0 SAFDIST .1 : Model loc d1 d2
NC:> DRILL DEPTH 1.0 SAFDIST .1 ENT $C1 $C2 ;
NC:> INDEX BAXIS ATANGL 90
NC:> DRILL : Model loc d1 d2
NC:> DRILL ENT $C3 $C4 ;
NC:> MOVE HOME
NC:> CHGTOOL 2 "CSINK"
NC:> CSINK DIA .438 : Model loc d1 d2
NC:> CSINK DIA .438 ENT $C3 $C4 ;
NC:> INDEX BAXIS ATANGL 0
NC:> CSINK : Model loc d1 d2
NC:> CSINK ENT $C2 $C1 ;
NC:> MOVE HOME
```

Figure 7-15 Rotary Table Motion for Hole Processing on a Cylinder



Displaying Machine Tool Motion

Use DISPLAY CYCLE to view machine tool motion.

Check the motion of your machine tool by viewing it with the DISPLAY CYCLE [SYS] command. DISPLAY CYCLE invokes and terminates the graphics emulation of machine tool cycles resulting from the following hole processing commands:

- DISPLAY CYCLE ON displays motion resulting from a hole processing command for the first location only. For subsequent locations, only motions to the approach and retract points are displayed.
- DISPLAY CYCLE ALL displays all motion resulting at all hole locations.
- DISPLAY CYCLE OFF ends cycle emulation for hole processing and subsequently displays only motion to APPROACH and RETRACT points.

Choose the version of DISPLAY CYCLE that best serves your current needs.

For example, to display the tool cycle for drill motion with the display mode turned on, enter

```
NC:> DISPLAY CYCLE ON
```

See the *CVNC System User Guide and Menu Reference* for more details about the DISPLAY CYCLE command.

Chapter 8

Tool Motion Generation: Noncutting

This chapter describes the noncutting commands, APPROACH, RETRACT, CLEAR, and MOVE, used to move the cutting tool between cuts.

- Overview of Noncutting Commands
- Positioning for a New Cut — APPROACH
- Withdrawing the Tool — RETRACT
- Moving to a Clearance Position — CLEAR
- Moving in Any Direction — MOVE

Overview of Noncutting Commands

Use APPROACH, RETRACT, MOVE, and CLEAR to move the cutting tool between cuts.

You can use APPROACH to move the tool to a new cutting location, which positions it on the ZAPPR plane.

With the CLEAR command, move the tool to a clearance plane (ZCLEAR).

To withdraw the cutting tool from its cutting location after completing a cut, use the RETRACT command. This brings the tool to the retraction plane, ZRETRACT.

You can use the MOVE command to move the tool in any direction.

Feed rates for the noncutting tool motion commands (APPROACH, RETRACT, MOVE, and CLEAR) are typically faster than feed rates of the cutting commands. The default for each is RAPID.

You can set the feed rate for each of the noncutting commands with the FEED command; subsequently, when you enter these commands, each will move at its assigned feed rate. For more information, see Chapter 5.

If you have assigned different feed rates to each of the noncutting commands, move the tool by entering these commands in turn, rather than making a series of feed rate changes to a single command.

Positioning for a New Cut — APPROACH

Use APPROACH to position the tool for a new cut. This places the tool on the approach z-plane, ZAPPR.

Specify an APPROACH feed rate with FEED before you enter APPROACH. The APPROACH feed rate defaults to RAPID if you do not specify a feed rate value.

The z-coordinate of the destination is ZAPPR (defined by PLANE before you enter APPROACH), unless you expressly override this value with subsequent modifiers and getdata.

You can use getdata to indicate the new location or specify the coordinates in absolute or incremental values.

You can move the tool either to a position on the selected entity or to a point tangent to the selected entity on the same or opposite side of the entity as the bias point. You can also specify the direction of motion.

For example, to move the tool to an absolute position of x = 2.45, y = 1.60, and z = 0.0 on the FILTWO entity, enter

```
NC:> APPROACH XABS 2.45 YABS 1.60 ZABS 0.0 ON FILTWO
```

Withdrawing the Tool — RETRACT

You can use RETRACT to retract your cutting tool after it has made a cut.

Use RETRACT to withdraw the cutting tool at the RETRACT feed rate after it has completed a cut. This will bring it to the retraction z-plane, ZRETRACT.

The default feed rate for RETRACT is RAPID, but you can change this value with the FEED command.

You can use getdata to indicate the new location or specify the coordinates in absolute or incremental values.

You can move the tool either to a position on the selected entity or to a point tangent to the selected entity on the same or opposite side of the entity as the bias point. You can also specify the direction of motion.

For example, to retract the tool to an absolute position of x = 3.45, y = 5.60, and z = 8.0, enter

```
NC:> RETRACT XABS 3.45 YABS 5.60 ZABS 8.0
```

Moving to a Clearance Position — CLEAR

You can move the tool to a clearance plane with the CLEAR command.

CLEAR moves the tool to a clearance location at the CLEAR feed rate. The z-coordinate of the destination is ZCLEAR (defined by PLANE you enter CLEAR), unless this value is expressly overridden.

Specify a CLEAR feed rate with FEED before you enter CLEAR. The CLEAR feed rate defaults to RAPID if you do not specify a feed rate value.

You can use getdata to indicate the new location, or specify the coordinates in absolute or incremental values.

You can move the tool to a position on the selected entity or to a point tangent to the selected entity on the same or opposite side of the entity as the bias point. You can also move the tool to the home point position specified by the HOMEPT command.

For example, to move the tool to a clearance location with an incremental position of x=1.80, y=2.45, and z=4.0, enter

```
NC:> CLEAR XINC 1.80 YINC 2.45 ZINC 4.0
```

Moving in Any Direction — MOVE

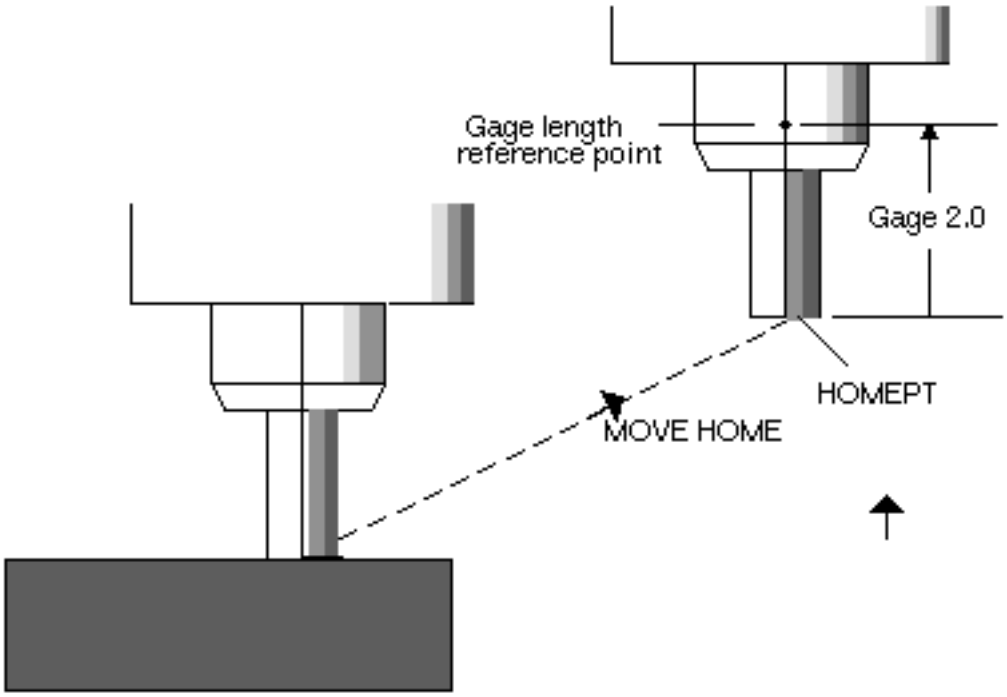
MOVE moves the tool (in any direction) from its current location to the new one at RAPID traverse.

You can use getdata to indicate the new location or specify the coordinates in absolute or incremental values.

You can move the tool either to a position on the selected entity or to a point tangent to the selected entity on the same or opposite side of the entity as the bias point. You can also move the tool to the home point position specified by the HOMEPT command.

The following example shows how this command can be used:

```
NC:> MOVE HOME
```



Glossary

ACS

Active Construction Space.

boring

Using a single- or multipoint tool to create a highly cylindrical and concentric hole.

CAD, Computer Aided Design

Product design done with the aid of computers and specialized software.

CAM, Computer Aided Manufacturing

Use of computers in manufacturing or machining operations.

center-drilling

Drilling tapered holes for mounting work between centers.

chamfering

Machining an angle or bevel on the shoulders of a workpiece for clearance or for the cutting edges of a tap.

CIM, Computer Integrated Manufacturing

Use of interconnected computers that control all phases of production, from management through sales, order processing, and chip-making.

circular interpolation

Taking a circle or parametrically described arc and approximating it with straight line segments to a specified tolerance.

CLFile, Cutter Location File

An output file generated from a JCF (Job Control File) that contained CVNC user input commands.

CNC, Computerized Numerical Control

Use of a microprocessor controller mounted on or accessible to the machine tool that permits creating or modifying of part programs created off-stream.

command files

Text files containing a sequence of NC commands that perform repetitive functions such as moving to the home point or changing the tool.

counterboring

Using weights to balance a workpiece, grinding wheel, or other rotating tool or workpiece for a smooth machining operation without vibration.

countersinking

Cutting an angled enlargement of a bore or hole to mount an angled screw head flush with a workpiece surface.

CPL

(1) A predefined construction plane (cplane). (2) A command that specifies a new active construction plane.

cutting tools

Any device used for making, cutting, grooving, shaping, reaming, boring, or removing, metals or materials.

CVNC-M2

Software that produces control tapes for 2- and 2 1/2-axis NC (numerical control) milling machines.

DNC, Direct Numerical Control

Operation of a machine tool using a series of programmed numerical data that activates the motors on a machine tool. Can be done using tapes or direct input from a computer.

drilling

Operation where the blank of the cutter is twisted to create a fluted shape, which creates round holes in a workpiece. Normally the first step in machining operations such as boring, reaming, tapping, counterboring, countersinking, spot facing, and center drilling.

grammar files

Binary files containing grammar, macros, and APT/CLFile pass-through statements; define the CVNC-M2 language.

home point

Location of the starting point for tool motions.

indexing

Reorienting the tool into a new position, usually by changing the z-axis.

JCF, Job Control File

A text file containing NC (Numerical control) commands and macros, core system commands, and output pass-through statements.

macro commands

Give you the ability to do logic programming, such as control, branching, and looping functions in your JCF.

modal parameter

A value that can be set in one command and automatically applied whenever that command is invoked. For example, a feed rate can be set for the CUT command by the FEED command, after which the CUT feed rate will always be the same whenever CUT is invoked. You can override a modal parameter in any single instance that you want to, or you can reset it.

offset

(1) The distance from a defined position. (2) A specified amount of stock (for example, .5 inch) that should be left on a part before the final milling to help ensure a result within the tolerances.

pass-through statements

Postprocessor commands included in the CVNC grammar files that can be entered like CVNC commands in your part programs. CVNC does not process them. Instead, they pass through to the APT, CLFile, or COMPACT II output file.

point macros

Macros referenced at predefined points. If a macro for a predefined point is present in the active macro libraries, then it is executed every time that point is reached; if a macro is not present, no action is taken.

slaved

If you have two rotary devices one can be slaved (mounted) to the other device of the same type. The slaved device is secondary and the device it is slaved to is primary. When the primary rotary device rotates, the slaved rotary device rotates around the primary device.

system variables

Machining and system parameters that result from the execution of CVNC commands in the JCF.

z-plane

A plane perpendicular to the tool axis, located a specified distance from the active CPL. Z-planes are used to cut to, plunge to, approach to, clear to, and retract to. There is also a z-plane used as a reference planes for other z-planes.

Index

A

AAXIS table machine
  configuring 2-18
Accessing
  CVNC-M2 1-7
Active construction space 1
APPROACH command
  procedure 8-3
Area clearance
  defaults 6-36
  operations 6-38
  tool path 6-35
AREAMILL (Area Milling) command
  boundaries 6-38
  contouring pass 6-39
  cutting sequence 6-38
  overview 6-35
  point macros 6-44
  positioning 6-39
  predefined points 6-36
  procedure 6-36, 6-37
  with
    QUERY modifier 6-43
    RETRACT modifier 6-41
AVOID modifier
  with hole processing commands 7-23
Avoidance parameters
  setting 7-23
Axis
  specifying rotation 4-10

B

Base system commands
  functions of 1-8
BAXIS table machine
  configuring 2-19
Bias points
  in CUT command 6-19
Blind and through-holes 7-13
BORE command
  procedure 7-9
Boring
  definition 1
  methods 7-10
Boundary
  defining 6-37
  machining
    around contiguous boundaries 6-25
    within closed boundaries 6-31

C

CADDS
  valid entities 1-2
CALCRAD (Calculate Tool Radius) command
  procedure 5-26
CAXIS 2-17, 2-22, 2-23, 2-25
Center-drilling
  definition 1
Chamfering
  definition 1
  with CSINK command 7-12
Check

entity 5-13
CHGTOOL (Change Tool) command
HOMEPT and 2-6
procedure 4-3
Circular interpolation 6-11
definition 2
CLEAR command
feed rate 8-5
procedure 8-5
Clearances
control 7-19
plane 8-5
CLFile
definition 2
ORIGIN statement 4-21
CNC (Computerized Numerical Control)
definition 2
Command files
definition 2
explanation 1-5
Commands
CVNC-M2 1-8
noncutting 8-2
Compiling macros 6-45
Compound
heads 2-23
tables 2-26
CONFIG (Configure) command
characteristics 2-32
HOMEPT command and 2-11
procedure 2-8
setup 2-2
Configurations 2-1
invalid 2-33
unsupported 2-33
Configuring
4-axis
milling machines 2-11
with rotary table 2-17, 2-20, 2-22, 2-25, 2-32
5-axis
head and table 2-29
milling machines 2-23
with rotary table 2-27
AAXIS and BAXIS head machines 2-15
compound heads 2-23
machine tools 2-8
machines 2-2
with HZERO modifier 2-30
Construction planes 4-4
Contouring
boundary 6-37
final pass 6-40
initial and final passes 6-39
COOLANT command
procedure 5-12
Coordinate system
defining linear 2-3
Counterboring
definition 2
Countersinking
defining tool for
restriction on 3-8
definition 2
with CSINK command 7-12
CPL (Construction Plane) command
procedure 4-4
system-generated 4-15
Cplanes
defined by DATUM command 2-2
definition 2
generating 7-24
specifying new 4-4
z-planes and 5-4
CSINK (Countersink) command
controlling hole depth 7-18
procedure 7-12
CUT ARC command
procedure 6-11
CUT CHECK command 6-20
CUT command
bias points in 6-19
functions of 6-2
normalcy points 6-17
procedure 6-4
working with 6-17
CUT ENTITY command
procedure 6-12
Cutting
circular 6-4
entity 6-12
linear 6-4
operations 4-2
sequence 6-38
Cutting tools
definition 2
CVMAC macros 1-5
compiling 6-45
link file 6-45
linking 6-45

CVNC-M2 1-2
  accessing 1-7
  command functions in 1-8
    JCFs and 1-11
  milling commands 6-2
CVNC-M5
  4-axis
    configuring 2-11
  5-axis
    configuring 2-23
CVNC-supplied macros
  DPOCK 6-34
  DPROF 6-29
  SPROF 6-30
Cylinder
  machining 7-24

D

DATUM command
  procedure 2-3
  rotary axes and Cplane defined by 2-9
  setup use of 2-2
Defaults
  area clearance 6-36
  Z-Planes 5-5
DEFINDEX (Define INDEX Defaults) command
  explanation 4-19
DEFTOOL (Define Tool) command
  procedure 3-4
DELTOOL (Delete Tool) command
  procedure 3-12
Depth
  control 7-15
  incremental values 6-29
DIACOMP (Diameter Compensation) command
  contact point output 5-22
  procedure 5-22
Diameter compensation register 5-22
DISPLAY CYCLE command
  procedure 7-26
DNC (Direct Numerical Control)
  definition 3
Documentation, printing from Portable
    Document Format (PDF) file xviii
DPOCK macro
  procedure 6-34
DPROF macro
  procedure 6-29
DRILL command
  procedure 7-4
  with NOANGL modifier 7-17
Drill point definition 3-7
Drilling
  center
    definition 1
  deep-hole and pause 7-5
  definition 3
Drive
  entity 5-13
DWELL modifier
  with hole processing commands 7-22

E

Entering 1-7
Entities
  extensions 6-18

F

FEED command
  procedure 5-6
Feed rates 5-2
  defaults 5-6
  modal 5-6
  slowdown 5-7
  specifying
    command for 5-6
Fillets
  in stock offsets 5-16, 5-17, 5-19
Five-axis machines
  configuring 2-23
Four-axis machines
  configuring 2-11

G

Gage reference lines
  rotary axes and 2-12
getdata

in noncutting commands 8-3, 8-4, 8-5, 8-6
Grammar files
definition 3

H

Hole
blind 7-13
depth control 7-15
through 7-13
Hole processing 7-2
avoidance parameters 7-23
boring 7-9
clearance distances 7-19
countersinking and chamfering 7-12
cylindrical parts 7-24
drilling 7-4
dwell time 7-22
hole depth 7-15
milling order 7-21
operations locations 7-21
tapping 7-13
tool motion display 7-26
Home point
definition 3
HOMEPT (Home Point) command
CHGTOOL and 2-6
CONFIG command and 2-11
procedure 2-5
setup use of 2-2
use before tool change 2-6
HZERO modifier to CONFIG
coordinate data of tool output by 2-30
example of use 2-31

I

INDEX command
to a Cplane 4-13, 4-21, 4-24
to a rotary axis 4-10, 4-15, 4-18, 4-20
validating 4-18
Indexing
definition 3
Indexing (programming)
rotary devices 4-8
rules 4-8
Islands
defining 6-37

J

JCF 1-4
JCF (Job Control File)
definition 3
JCFs 1-4
configuration information in
job setup 2-2
tool definition 3-2
tool library for 3-3
generating and working with 1-11
Job Control File (JCF)
definition 3
Job setup
commands overview 1-9
machine configuration
commands ordering in 2-2
coordinate system and part program zero 2-3
home location 2-5
initial setup 2-2
rotary axis definition 2-8
tool definition 3-2
geometry specification 3-4
library setup 3-3
parameter listing 3-12
tool deletion from library 3-12

L

Lace cut
boundary 6-37
moves 6-40
with RETRACT modifier 6-41
Libraries
tools 3-3
Linear motions 6-4
Link file 6-45
LISTOOL (List Tool Parameters) command
procedure 3-12

M

Machine tools 2-1
Machines
  configuration 2-2, 2-8
  control statements 6-43
Machining
  around entities 6-25, 6-37
  closed boundary 6-31
  profiles 6-29, 6-30
Machining parameters 4
Macro commands 3
Macros 1-5
  definition 3
MILL7 modifier to DEFTOOL
  2-parameter tool definition with 3-9
  restriction with countersink tools 3-8
Milling
  CVNC-M2 commands 6-2
Modal feed rates 5-6
Modal parameters
  definition 3
Motion commands
  CVNC-M2 1-10
MOVE command
  procedure 8-6
MULTAX (Multiaxis Machining) command
  ON, affect 4-28
  procedure 5-29
Multiaxis output 5-29

N

NC process
  description 1-2
NCGROUP (Entity Grouping) command
  created by AREAMILL command 6-35
  with
    hole processing 7-21
    POCKET command 6-31
    PROFILE command 6-25
Noncutting motion
  commands for 8-2
    clearance positioning (CLEAR) 8-5
    moving a tool (MOVE) 8-6
    positioning cuts (APPROACH) 8-3
    withdrawing a tool (RETRACT) 8-4
Numerical control
  computerized (CNC)
    definition 2
  direct (DNC)
    definition 3

O

Offsets
  definition 4
  stock 5-13
    fillets specification 5-16
  tool 5-26
Operation setup
  commands overview 1-9
  cutting operations regulation 4-2
    coordinate systems (I/O)
      relationships 4-29
    Cplane selection 4-4
      in indexed plane 4-27
    rotary axis programming 4-8
    tool selection 4-3
  CVNC to NC machine relationships 4-33
  operational parameters setup 5-2
Operational parameters 5-2
  coolant control 5-12
  digital compensation register 5-22
  feed rates 5-6
  multiaxis output 5-29
  reference planes 5-2
    setting 5-4
  spindle speeds 5-11
  stock offsets 5-13
  tolerances (circular) 5-21
  tool offsets 5-26
  z-planes 5-2
    setting 5-4
Output
  CVNC-M2 1-5
  generating
    with point macros 6-45
  pass-through statements 1-6
Output macros
  referencing 6-44
  with AREAMILL command 6-36

P

Parameters
  avoidance 7-23
  modal
    definition 3
  operational 5-2
  system 4
  tool
    listing 3-12
    retrieving 3-7
Parameters, setup 5-2
Pass-through statements
  CVNC-M2 and 1-6
  definition 4
PLANE (Z-plane Setup) command
  procedure 5-4
PLUNGE command
  functions of 6-2
  procedure 6-24
POCKET command
  pinched-off 6-32
  procedure 6-31
Pocket cuts
  specified depth 6-34
Point macros
  definition 4
  procedure 6-44
Positioning 6-41
Printing documentation from Portable Document Format (PDF) file xviii
PROFILE command
  procedure 6-25

R

Reference planes 5-2
RETRACT command
  procedure 8-4
Rotary
  axes
    DATUM Cplane and 2-9
    programming 4-8
  device programming (INDEX) 4-8
  heads
    using HZERO modifier 2-30
  table
    configuring 4-axis 2-17
    table, AAXIS 2-18
Rotary axes
  gage reference lines and 2-12
Rotating
  around axes 4-10
  machine 4-18, 4-20

S

SAFDIA modifier
  clearance diameter 7-19
SAFDIST modifier
  clearance distance 7-19
Setup
  job setup
    machine configuration 2-1
    tool definition 3-1
  operation setup
    cutting operation regulation 4-1
    operational parameters 5-1
  operational parameters 5-2
  tool libraries 3-3
Side-approach move 6-39
Slaved (mounted)
  definition 4
SPEED command
  procedure 5-11
Spindle speed 5-2
  command for 5-11
SPROF macro
  procedure 6-30
Stock
  incremental values 6-30
  offsetting 5-13
    fillet specification in 5-16
STOCK (Machining Stock) command 5-13
  example 5-13
  offsets 5-13
System
  commands, overview 1-8
  monitoring tools 1-5
System variables
  CVNC-M2
    explanation 1-4
  definition 4

T

TAP command
  procedure 7-13
Tap thread definition 3-7
Through-holes 7-13
THRU modifier
  controlling hole depth 7-15
TLIB (Tool Library) command
  procedure 3-3
TOLER (Tolerance) command
  procedure 5-21
Tolerances 5-21
  specifying 5-21
Tool axis vectors 5-29
Tool motion 1-10
Tool paths
  lace cut 6-35
Tools
  axis planes 5-4
  axis vectors 5-29
  changing 4-3
  coordinate system for 2-3
  countersink 3-8
    restriction on 3-8
  defining 3-4
    2-parameter 3-9
    ends of 3-4
    geometries for 3-4
  deleting 3-12
  diameter compensation 5-22
  ends of 3-4
  feed rate specification 5-2
  hole processing operations 7-2
  libraries
    deleting tools from 3-12
    setting up 3-3
  listing parameters for 3-12
  milling operations 6-2
  motion
    commands overview 1-10
    displaying 7-26
    generating 1-10
  moving between cuts 8-2
  noncutting motion 8-2
  offset
    calculating 5-26
    offsets 5-26
  radius calculation 5-26
  redefining 3-12
  retracting 8-4
  selecting 4-3
  selection 4-3
  spindle speed specification 5-2
  withdrawing 8-4

V

Vertical milling machine 2-18

Z

Z-planes
  defaults 5-5
  definition 4
  setting 5-4
  types of 5-4